CERN – EUROPEAN ORGANIZATION FOR NUCLEAR RESEARCH

CLIC – Note – 1073

FINITE ELEMENT MODEL FOR THERMAL-STRUCTURAL ANALYSIS OF CLIC LAB MODULE TYPE 0#2

Antti Moilanen[1,2], Markus Aicheler[1,2], Alex Vamvakas[1], Jukka Väinölä[1,2], Steffen Doebert[1]

[1] CERN, Geneva, Switzerland
[2] Helsinki Institute of Physics, Finland

Abstract

Temperature changes lead to unwanted thermo-mechanical deformations in the components of the Compact Linear Collider (CLIC) module. There are several sources and sinks of heat around the CLIC two-beam module. Heat is generated in the components that produce, transfer, and extract radio frequency (RF) power. Excess heat is removed from the components by cooling water as well as dissipated to air by convection from the outer surfaces of the components. The ambient temperature might also vary along the tunnel during the operation of CLIC. Due to tight assembling and alignment tolerances, it is necessary to minimize the thermo-mechanical deformations in the components. In this paper, the steps of thermal-structural Finite Element Analysis (FEA) of CLIC lab module type 0#2 are described from geometry model simplification to setting up the simulation. The description is accompanied by useful hints for CATIA and ANSYS users performing similar modelling tasks. A reliable computer simulation is important for studying the effects of different operational parameters as well as evaluating future two-beam module designs.

Geneva, Switzerland
11.2.2017

Contents

1. Introduction

The Compact Linear Collider (CLIC) two-beam acceleration scheme relies on using the electromagnetic field from the high-current and low-energy Drive Beam (DB) to accelerate the low-current and high-energy Main Beam (MB) [1]. The CLIC is designed to be built in compact few-meter modules that are manufactured on ground and transported into tunnel underground. Each two-beam module consists of numerous components: radio frequency (RF) structures such as power extracting and transfer structures (PETS), super accelerating structures (SAS), drive beam quadrupole magnets (DBQ), and compact loads (CL). In addition there are waveguides connecting the two beams, vacuum pumps, sensors, actuators, girders for both main and drive beam, supports for the components, and so on. During operation of the accelerator, heat production in the RF structures is compensated by a water cooling system. Heat is also dissipated to air by convection from the outer surfaces of the components.

Due to tight assembling and alignment tolerances of the RF structures in the CLIC module, it is important to study the thermo-mechanical deformations arising from thermal gradients, which originate from heat production in the RF structures, cooling, as well as changes in ambient temperature. One way to study the displacements of different components is to conduct experimental tests on prototype modules built in the laboratory. To save time and resources, thermo-mechanical deformations can be studied using simulation methods such as Finite Element Analysis (FEA).

Computational resources required by FEA depend on type of the analysis (transient or steady-state thermal, static or dynamic structural, etc.), complexity of the analysis (linear or non-linear, include contacts or not, etc.), and number of elements. In this study, we combine the solutions of two analysis types in ANSYS Workbench: Steady-State Thermal and Static Structural. Both types are relatively simple compared to, for example, time-dependent transient analysis. The number of elements can be reduced by removing components from Computer Aided Design (CAD) model, suppressing geometric details that are not relevant for the study, and adjusting the mesh size.

In this paper, finite element model for thermal-structural analysis of CLIC lab module type 0#2 is described step-by-step. The work is based on previous studies of CLIC two-beam module type 0#1 [2, 3, 4, 5, 6]. Both of the module types are currently built as mechanical mock-up prototypes in our laboratory. The main differences between the module types 0#1 and 0#2 are removal of the central vacuum tank, changing the mock-up DBQ magnets to real ones, and using 4 SAS (i.e. two AS joined together by brazing) instead of one two-metre long solid structures consisting of 8 brazed AS. The paper is intended to serve as a guide for people working on thermal-structural analysis using ANSYS and especially with future two-beam module designs. This article is structured as following: In Chapter 2, steps of model simplification and creating the cooling system are described. In Chapter 3, the necessary settings for thermal-structural FEA are listed. The paper closes with conclusions and outlook for future work in Chapter 4.

2. Preparation of geometry

Obtaining sufficiently accurate results with minimum number of elements requires simplifying the geometry. The CAD model of the CLIC module is relatively heavy with all the components involved (PETS, SAS, DBQ magnets, RF structures, vacuum pumps, cooling system, sensors, girders, supports, adjustment devices, flanges, bellows, screws, bolts, nuts, spheres, and so on). For example, the complete CAD model of the CLIC lab module type 0#2 consists of more than 1000 parts. Moreover, the parts contain a lot of fine details such as chamfers, rounds, gaskets, and holes.

Preparing the geometry is started with model simplification, which includes removing components from the CAD model and suppressing geometric details. Aside with model simplification, a cooling system is created and included in the geometry before importing to ANSYS Mechanical for FEA.

2.1 Model simplification

Model simplification is done in two phases: first, some unnecessary components are deleted or deactivated in CATIA (version V5). Then, more parts and geometric details are suppressed in ANSYS (version 17.1) geometry editor: DesignModeler (DM) or SpaceClaim Direct Modeler (SCDM). The steps of component and geometry-detail removal performed in each software are presented in *Table 1*. The details of these steps are discussed in the following sub chapters. Note that this workflow for model simplification is just one of many possibilities. For example, all model cleaning and preparation could be done in CATIA. Here the idea is to showcase and discuss the possibilities for model simplification in different software as the best compromise between efficiency and flexibility.

Table 1. Steps of model simplification.

Step #	Software	Action
1.1	CATIA	Delete components not included in the analysis
1.2	CATIA	Deactivate components not included in the analysis
2.1	ANSYS DM	Suppress components possibly included in the analysis later
2.2	ANSYS DM/SCDM	Remove small and unnecessary geometric details

Building up the finite element model is started by opening the CAD model of the CLIC module. At the time when this CLIC note is released, the CAD model of the CLIC module was developed using CATIA and stored in the CERN mechanical design and document managing system SmarTeam. After opening the model in SmarTeam, it is advisable to save a copy of it locally to the user's personal computer to avoid any unwanted changes in the SmarTeam documents.

Step 1.1: Component deleting in CATIA

Unnecessary components are deleted directly from the CAD model in CATIA. As a guideline for thermal analysis, most of the components that are not likely to have a notable effect on thermal distribution can be removed. For example, all the standard mounting parts such as screws and bolts can be deleted. Note that whereas in thermal analysis the components could "float" in the air without being attached to anything, in structural analysis the components must be in contact with another component by at least one surface or fixed by a support.

Deleting components makes the CATIA model lighter to load and use. In addition to deleting the components in CATIA, the corresponding .CATpart and .CATproduct files can be deleted from the local folder to save hard disk space.
Hint: For deleting the parts from your PC, it is useful to keep a list of the names of the deleted components and assemblies as you delete them in the CAD model. It is good to keep in mind that

certain type of component may appear in multiple places around the module, and if the files are deleted too early from the local folder, the CAD model is not able to retrieve those parts anymore and might prevent loading the model correctly. Thus, delete the files from the local folder only after all the component-deleting from the CAD model is done.

Step 1.2: Component deactivation in CATIA

Hint: if you are sure about the components that are going to be included in the analysis, you can directly delete all the unnecessary components and skip the step 1.2.
Sometimes it may be useful to deactivate (in CATIA, this operation is called Inactivate) some of the components instead of deleting them permanently. Note that even though the deactivated components are not visible in the geometry, they still remain as part of the CAD model. Thus, the corresponding files have to be preserved in the local folder as well.

After performing steps 1.1 and 1.2 in CATIA, the model can be saved as .CATProduct or .stp file for importing to ANSYS. Saving as .CATProduct maintains the assembly file that calls the .CATPart files from the local folder, making it perhaps easier to replace single components with updated ones later. Saving as .stp file renders the geometry to a single package containing all the parts in one file.
Hint: It is a matter of opinion which geometry file format is the best for importing in ANSYS. In this work, the file was imported as .CATProduct. Note that deactivated components are not included in the .stp file in ANSYS, whereas they are in the .CATProduct file.

Step 2.1: Component suppressing in ANSYS

The reduced CAD model is imported in ANSYS geometry editor. In practice, the CAD model of the CLIC module includes so many parts that it is very likely to find some more unnecessary components such as pins and screws at this stage of model simplification. Fortunately, components can still be suppressed in ANSYS. Suppressing makes the components invisible and excludes them from the FEA as well. Hide operation can be used instead of Suppress for just hiding the components from the visual appearance, having them still included in the FEA.

The good thing about suppressing components is that they can easily be unsuppressed later to include them back in the analysis. The downside of suppressing components in ANSYS is that similar type of components must be suppressed separately. For example, if a certain standard type of screw is deleted or deactivated in CATIA, the software usually recognizes links to other identical screws in similar locations around the model and performs the action to all of them. In ANSYS all the screws would have to be selected separately.
Hint: Named Selection is a very useful tool for suppressing multiple similar components from the model, as you can suppress all the parts in the Named Selection at once.

Step 2.2: Geometric details removal in ANSYS

The model should now include only the components that are necessary for the analysis. The parts have a lot of geometrical details such as small holes, chamfers, overhangs, and labels which may not have notable effect on the thermal distribution, but would require a dense mesh around them and hence increase the number of elements in the analysis. For example, removing pins and screws leaves behind holes that can be removed because their effect on thermal and structural behavior is (usually) negligible.

In ANSYS Workbench it is possible to choose the default geometry editor by selecting: Tools – Options – Geometry Import – Preferred Geometry Editor. In version 17.1 there are two available geometry editors: the conventional DesignModeler (DM) and more recently added SpaceClaim Direct Modeler (SCDM). In this study, both geometry editors were used for model simplification and creating cooling system. Note that there is no possibility of suppressing parts in SCDM; only permanent deleting. Nevertheless which geometry editor is used, parts can always be suppressed and unsuppressed later in ANSYS Mechanical when setting up the analysis.
Hint: Both geometry editors include a variety of tools and automatized functions for model simplification. However, the features vary a lot between the editors so it is worth spending some time checking out what is available in each.

ANSYS DM provides many tools for automatic cleaning of the model. For example, the Repair holes function is useful for removing holes from a body automatically. The Repair holes function does not always succeed in removing all of the holes. The reason for failure is usually too complex geometry, e.g., involving several overlapping holes and inclined or non-planar surfaces. The function works best on holes through a body with parallel top and bottom surfaces. Repair holes usually works also for grooves on surfaces. In case of failure, one can always perform manual cleaning by removing holes with Fill and chamfers and rounds using Face delete.

Note that after filling holes manually, the newly created bodies need to be merged to the main body using Boolean operation Unify. The Repair holes function takes care of this operation automatically, but it does not always repair all the holes so manual inspection is recommended in any case. For Repair holes, it is possible to define the minimum and maximum sizes of the holes to be removed.
Hint: It is a good practice to rename the operations in the ANSYS DM operation tree with corresponding component names or such. This helps you (and especially someone using the model after you) to keep track on the operations done.
Hint: From the experience of the author, preserving and generating the operations again and again in ANSYS DM makes it crashing or not responding more often. Modifying an operation from the past can also lead to significant mismatch errors with the following operations.

ANSYS SCDM is a flexible tool for handling geometries as it treats the CAD model as if it was a piece of wax. One can, for example, grab by a corner of the component and simply stretch the piece in one or more directions. There exists no operation tree such as in the ANSYS DM, so modifications cannot be made to operations performed several steps before. On the other hand, there is neither Generate button to click each time after an operation, which already makes SCDM significantly lighter to use than DM. Both geometry editors allow for basic Undo operation of the consecutive previous steps. SCDM is lighter also in performing many operations. For example, holes, edges, and surfaces can simply be selected and deleted. More information on the properties of each geometry editor can be found from the ANSYS help and documentation.

2.2 Modelling cooling system

In CLIC lab module type 0#2, heat production in the RF structures is compensated by water cooling. The cooling system covers all 4 SAS, 2 double-PETS, 20 CLs, 4 waveguide cooling tubes, and the coils of the 2 DBQ magnets. Cooling water runs through hollow copper wires in the DBQ coils removing the heat directly from its source and thus making the cooling very efficient. Therefore, we can approximate that most of the heating power from the coils is taken away by water and so the net heating power on the yokes and supports is negligible. In our model, the DBQ coils and thus also the respective cooling circuits are left out of the analysis, because meshing the hollow copper wiring would result in very large number of small elements. In the future, also the DBQ cooling should be modelled to improve simulation accuracy. This could be done by, e.g., replacing the real coil geometry by a piece of solid equipped with a single effective cooling channel.

Modelling the cooling system is started by sketching lines and curves along the cooling channels and tubes. The cooling network is drawn in five individual circuits: one circuit for each SAS with the respective surrounding CL, and one unified circuit for the PETS with the CL located above the MB side. The complete cooling system with these 5 circuits are highlighted with yellow in *Figure 1*.
Hint: In SCDM, a useful tool for ensuring that all the lines and curves are connected is found via path Repair → Curve Gaps.

It is essential to check the edge directions of the cooling circuits, because these will define the direction of cooling water mass flow. To do this choose View – Display Edge Direction and inspect the cooling circuits for any discontinuities. The edge directions indicated by arrows in *Figure 2* can be reversed in the geometry editor. In ANSYS DM this is done by clicking an edge and changing the Reverse Orientation option.

After drawing the wireframes and making sure there are no gaps or discontinuities in the system, the parts are assigned as beams with circular cross section. The cross section is created via path Concept → Cross Section → Circular in ANSYS DM. The radius of the cross section in this model was 0.003 m. Note that this only affects the geometrical appearance of the beam that representing the water. The actual effective radius for the water flow is taken into account later in the boundary condition settings (Mass Flow Rate).
Hint: In SCDM, one can extract beams out of existing components such as pipes, thus avoiding drawing them manually. The tool can be found via path Prepare → Extract. SCDM provides also other flexible tools for drawing this kind of wireframe geometry: for example fast modification operations such as Pull and Move. In this work, the wireframes for PETS and SAS were drawn separately in SCDM and imported in DM as new parts. Then, the edge directions were inspected in DM after realizing their importance for the mass flow in the analysis.

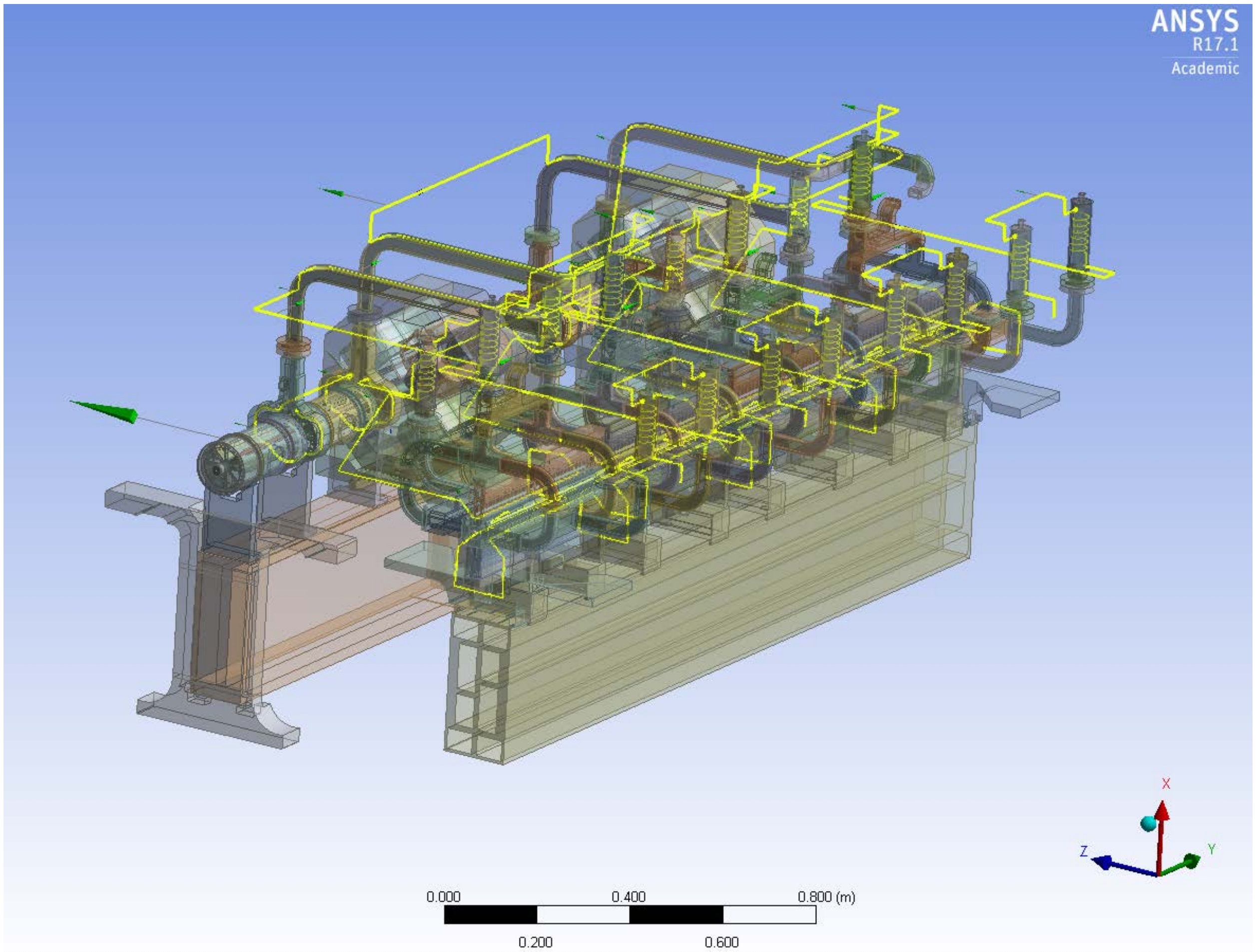


Figure 1. Simplified geometry model of CLIC lab module type 0#2. Cooling water network is highlighted with yellow.

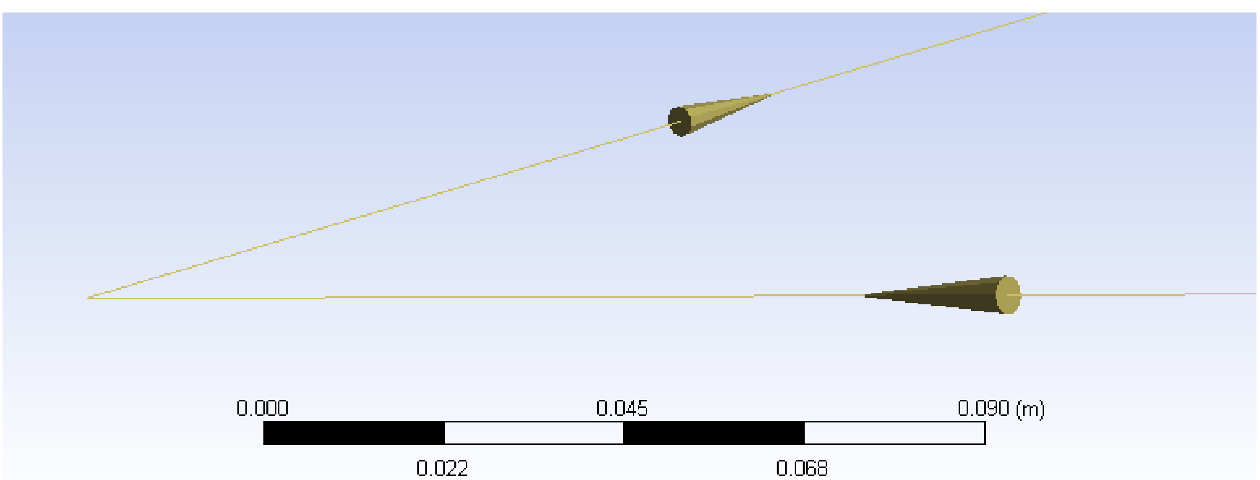


Figure 2. Edge directions indicated by arrows define the direction for mass flow in the cooling network.

3. Thermal-structural analysis

After simplifying the model and creating the cooling system, the geometry is ready to be imported in ANSYS Workbench for thermal-structural analysis. A good practice is to import the model into a standalone Geometry module and link the geometry to the systems, as shown in *Figure 3*. This allows for linking the same geometry module for several analysis systems so that updating or changing the geometry file will propagate through all of the analysis systems in the chain.

Solution to Steady-State Thermal analysis is imported as load to the setup of Static Structural analysis. Engineering Data, Geometry, and Model are linked between the analysis systems as they are. If needed, the links can be destroyed simply by right-clicking and deleting, and new ones created by drag and drop. Input and output parameters are stored in a separate Parameter Set module. Having all the input parameters (initial and ambient temperatures, water inlet temperature, heat transfer coefficients, mass flows, heat powers, and temperature boundary conditions) in the Parameter Set makes it easier to run the simulation with different parametric cases. The Project Schematic of the combined thermal-structural analysis is shown in *Figure 3*. Next, we will go through the settings for our analysis in ANSYS Mechanical.
Hint: Changed parameters are not always updated automatically to the analysis. Then it is necessary to update input parameters manually by right-click and Refresh Project.

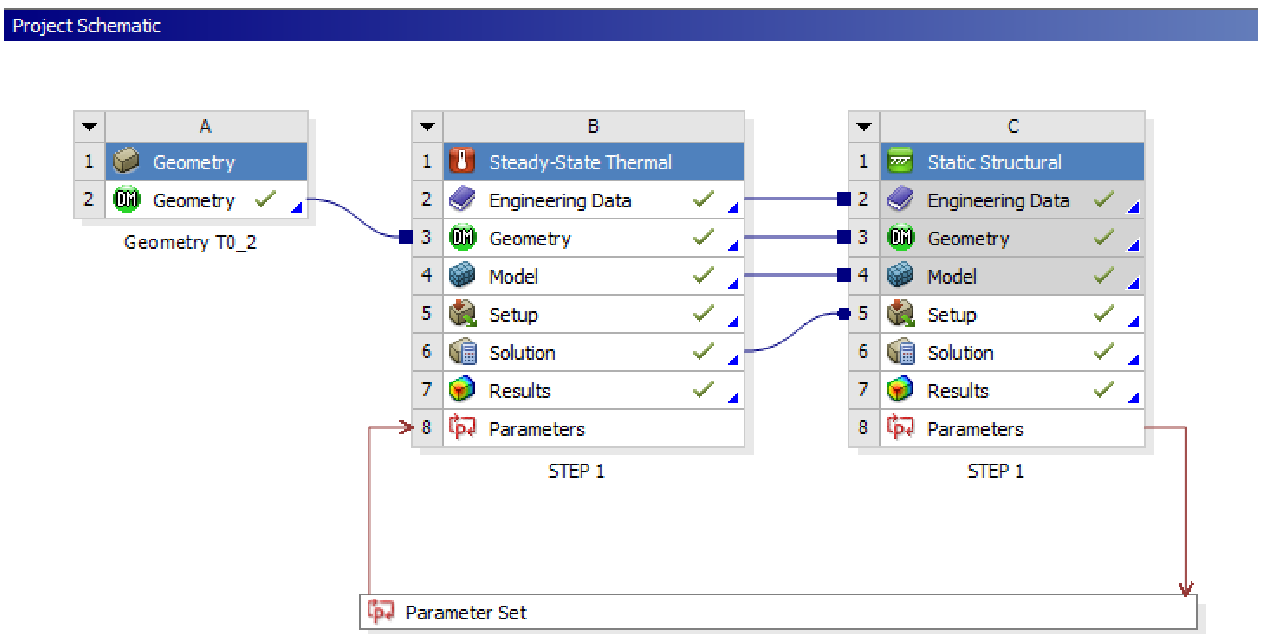


Figure 3. Project Schematic of the analysis in ANSYS 17.1 Workbench.

3.1 Geometry and Coordinate Systems

First thing to do after opening the model in ANSYS Mechanical is to assign materials for all parts. The MB and DB girders are made of silicon carbide, whereas most of the components such as SAS, PETS, waveguides, etc. are Oxygen-Free Electronic (OFE) copper. Flanges, DBQ frames, and covers around the PETS are stainless steel. Arms for wire sensors and support plates for DBQs are aluminum. For cooling circuits, material is chosen as water and Model Type is set to Thermal Fluid.

In real experiments, the displacements are tracked with the help of attached fiducials (standard metrological glass spheres) placed on the outer surfaces of girders and components of the lab module. Positions of the fiducials are measured using a laser tracker Leica AT401 [7]. In addition to tracking fiducials, optical and capacitive sensors are used to detect the location of a stretched wire acting as a reference straight line for components alignment.

In FEA, the deformations can be tracked by placing Coordinate Systems to the points corresponding to the locations of (the center points of the) fiducials. It is then straightforward to probe deformation of each tracked point for direct comparison with experimental measurements. Similar procedure can be done for the positions of thermal sensors to export surface temperatures as output results.

The fiducial spheres were suppressed from the geometry model, so the coordinates of the center points of the fiducials would float in the air when imported in ANSYS. Thus, to be able to probe the deformations, each fiducial coordinate point is shifted manually to the nearest point on a component or girder surface where the fiducial is attached to in reality. Thus, the deformation at the fiducial point is assumed to be equal to the deformation at the nearest point on the surface. Effectively this corresponds to a rigid connection between the surface point and the fiducial point, which is a valid assumption when there are no significant rotations involved. Note that in order for the Coordinate Systems to follow the deformation of the components correctly, they have to lie precisely on the surfaces.
Hint: The fiducial point coordinates can be imported as a .txt file to ANSYS Geometry editor. Shifting the coordinates to the surfaces is done manually by measuring the distance between the imported fiducial coordinate to the surface where the fiducial is attached in reality. Then the corresponding coordinate translation is done in Microsoft Excel, and the new point, now lying on the surface of a component, is imported in ANSYS Mechanical as a Coordinate System. Deformation Probe is set to this Coordinate System, and the results from the probes are exported as Output parameters. It is advisable to maintain the naming of the fiducials in both Coordinate Systems and Deformation Probes.

3.2 Connections

When a geometry is imported in ANSYS Mechanical, bonded contacts are automatically created between nearby surfaces. This results in connections between surfaces that are not even attached to each other in reality. It is therefore necessary to go through all the automatically created connections, and suppress or modify the non-realistic ones. In addition to the automatically created contacts, there are certain places in the model that require manually defined contacts. Below we list the contacts that were defined manually.
Hint: Due to large number of components, there are hundreds of automatically created bonded contacts around the module which are named generically as Contact Region 1, 2, 3, etc. A nice way to make the list more convenient is to group the contacts in folders named after the parts of the module where they appear.

The bellows between each SAS on the MB side, and between double-PETS and DBQs on the DB side were deleted from the model. The bellows in the lab module are assumed to behave so flexibly that no structural (nor thermal) effects are transferred through them.

It is possible to make a thermal contact between surfaces. The waveguides on top of the module are connected by an out-coupling bellow which was suppressed from the model. Instead, thermal contact is defined on the connected surfaces with pinball region large enough to cover both of the surfaces that are in contact with the bellow in reality. The pinball region is depicted in *Figure 4*. To activate only the thermal degree of freedom, the following command is inserted to bonded contact: keyopt,cid,1,2.

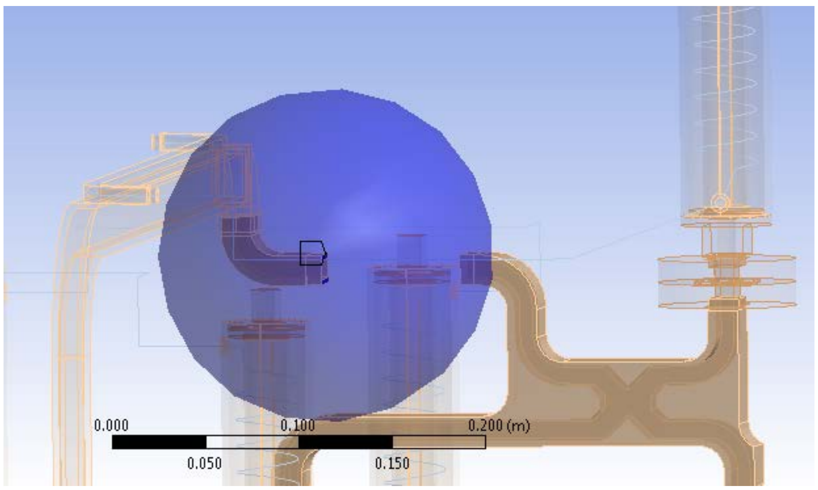


Figure 4. Illustration of the pinball region for bonded contact modeling the out-coupling bellow.

Bonded contacts are defined for all the contact surfaces between V-supports and SAS/PETS. In reality, the components are clamped on the supports but they can still slide along the support interfaces. It would be more realistic to use frictional contacts in some of the support interfaces.

The double PETS in the lab module type 0#2 were originally supported by three V-supports each. Later it was noticed that the structure does not need this additional constraint for minimizing sag and the third support for double PETS was physically removed. Thus, also in the simulation the contacts between PETS and the middle V-supports are suppressed.

3.3 Mesh

The mesh should be dense enough to be able to model the macroscopic deformations in the most critical parts of the module (e.g. SAS because it has the tightest tolerances), whereas being coarser in less important parts (e.g. girders because they have low thermal expansion coefficient so no significant deformations are expected). Often it is sufficient to create mesh with the automatic meshing functions in ANSYS Mechanical trying out a few different Relevance settings and inspect the mesh visually. Sweep method is recommended by previous simulation studies [3]. ANSYS Mechanical provides numerous tools for advanced mesh sizing and quality checking. In this analysis, Relevance 0 with Coarse Relevance Center was used along with Body Sizing having Element Size defined as 5 mm. This results in total 2 million nodes and 1.1 million elements.

3.4 Named Selections

Named Selections are created for fluid edges and convection surfaces that are in contact with the cooling water. These Named Selections are later called in boundary conditions defining the heat transfer between surface and fluid elements.
Hint: The number of surfaces to be selected for convection can be very large. Fortunately, all of them don't have to be selected in one go. Several Named Selections can be created first and then merged them into one single Named Selection by selecting right-click – Group.

3.5 Analysis settings and boundary conditions

Initial temperature is set to 20 °C, however this does not have any effect on the results in Steady-State Thermal analysis. All the available controls are set to Program Controlled. Heat Flows are defined for all the components in the module that generate heat power: 4 x SAS, 2 x DBQ, 4 x PETS, and 20 x CL. The heating powers of the components are listed in *Table 2*. Because we are studying a lab module without any running particle beams, the heating powers are mimicked by other means. For SAS, heater elements are placed in the grooves along the outer surfaces of the components, see *Figure 5*. For PETS and CL, the structures are wrapped around with heating elements, so in the simulation heating is set on the outer surfaces marked as green in *Figure 6*.

As discussed earlier, the coils of the DBQs are suppressed from the simulation, and the net heating power of the coils on the yokes and supports is assumed to be negligible. Nevertheless, we still need to take into account the temperature of the coils which for now stays constant. Based on the nominal parameters of DQBs with power of 556 W, inlet water temperature of around 30 °C, and water flow of 1.8 l/min (0.108 m^3/h) the resulting temperature gradient on the DBQ is 5 °C. Based on this, we also assume in the simulation that the cooling water temperature of the coils is increased by 5 °C and therefore place a temperature condition of 30 °C (the inlet water temperature is taken to be 25 °C, but this is anyway a rough estimation about the coil temperature) on the surfaces where coils are in contact with the yokes. Improving this model of the thermal effects of the DBQ coils requires further experimental measurements.

In addition to Heat Flows, two types of heat convection occur around the module: convection by air flow and convection by cooling water. Basically, all outer surfaces of the components are exposed to convection by air flow. Preliminary studies showed that the air speed and thus also the Film Coefficient (heat transfer coefficient) may drop down to 20 % in different parts around the module. However, in this work for simplicity and in lack of better data the heat transfer coefficient for air convection is taken to be constant in space. Calculating the theoretical value for heat transfer coefficient is done according to the theory summarized in previous works [3, 5]. With air flow speed of 0.7 m/s and at temperatures 10…50 ˚C the heat transfer coefficient for laminar parallel flow over a flat surface is around 4 W/(m^2 ˚C). The parameters for dry air at different temperatures were obtained from [9]. In the theory, there is a parameter called characteristic length of flow, which should somehow represent the basic dimension of the obstacle blocking the free flow of fluid. In this work, the characteristic length is taken to be one meter. Scaling the parameter will scale the heat transfer coefficient according to power of -1/2. Hence, varying the characteristic length between 0.01…1 m changes the heat transfer coefficient roughly from 40…4 W/(m^2 ˚C). It is important to keep in mind, that the theory presented in [3, 5] applies only for the very simplistic case of parallel flow over a flat surface. Thus, the theoretical value for the heat transfer coefficient should not be taken as a practical or real value for our module. A more elaborate modeling of the air convection could be done by fluid dynamics simulation of the air flow around the module using, e.g., ANSYS Fluent. Other approach would be to measure the air speed or heat flux on the surfaces around the lab module and to import the distribution in the simulation.

Heat is dissipated from components also by thermal radiation. In previous studies, the radiation was taken into account by adding a theoretically defined value of 6 W/(m^2 °C) to the convection heat transfer coefficient [10]. In our work, radiation is included as a boundary condition similarly to convection. In ANSYS 17.1 the radiation condition simulates thermal radiation from surface to its environment, whereas surface-to-surface radiation is not supported. Emissivity parameter describes the relative emissivity of the surfaces compared to an ideal black-body. In this work, the emissivity is set to 0.8 which is close to the usual values reported for copper, aluminum, and stainless steel.

The inner surfaces of the cooling channels are exposed to heat convection by water. Modeling fluid flow via thermal "pipes" has been a product feature since ANSYS version 16.0 [8]. In the previous versions, thermal convection between fluid and convective surfaces had to be defined using APDL command objects as done in [3, 5]. After inserting Convection to the boundary conditions, there is an option for Fluid Flow for which one must select the corresponding Fluid Flow Edges. This is done easiest by using Named Selections. Film Coefficient (i.e., heat transfer coefficient) for the Fluid Flow is calculated as presented in previous works [2, 3, 5]. The Mass Flow Rate is inserted to the boundary conditions as well.

Table 2. Dissipated heat powers of RF structures used in simulation.

Structure	Heat power (W)
SAS	820
PETS	110
CL	150

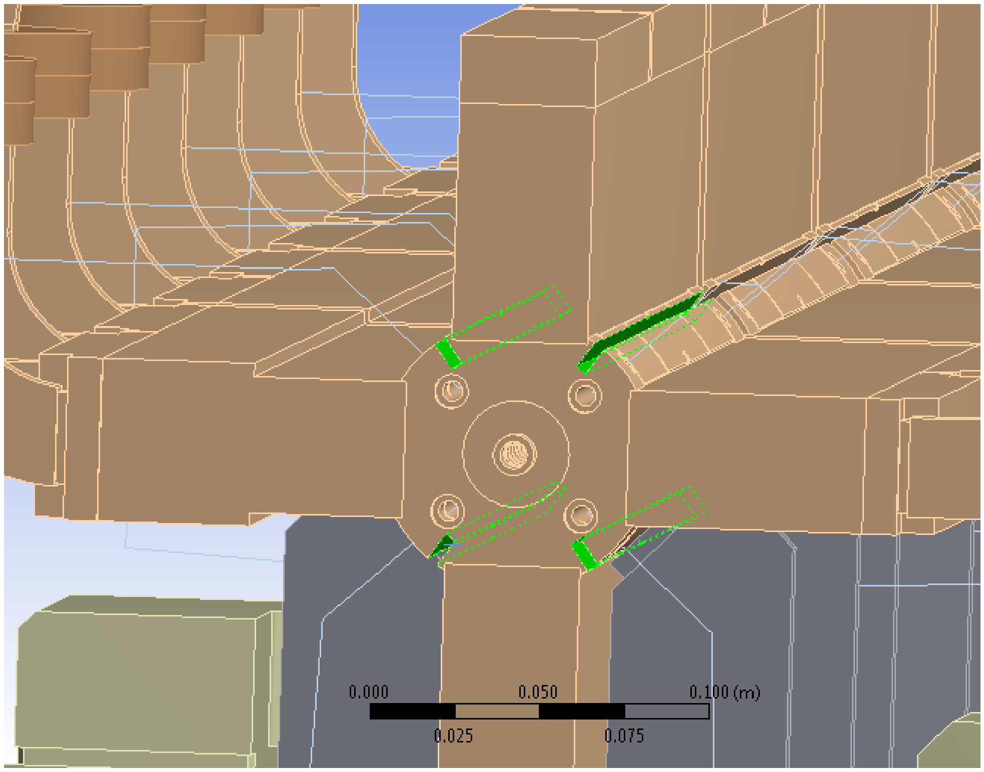
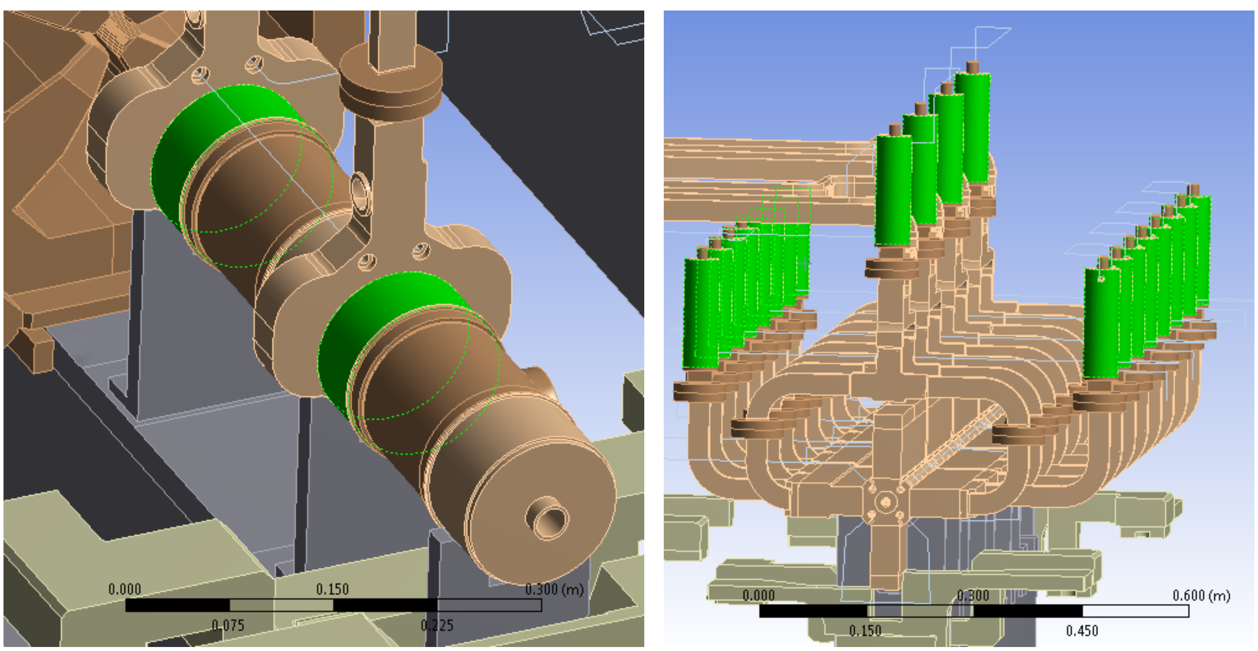


Figure 5. Heating of SAS in the grooves along the outer surfaces highlighted with green color.



(a) *(b)*

Figure 6. Heating of a) PETS and b) CLs on the outer surfaces highlighted with green color.

3.6 Structural analysis

The results of Steady-State Thermal analysis are transferred to a Static Structural analysis system in ANSYS Workbench. The temperature distribution is imported as a thermal load for all bodies besides the cooling system wireframes, which are suppressed from the structural analysis. Gravity is taken into account by imposing Standard Earth Gravity with vertical acceleration of 9.81 m/s^2. Fixed and frictionless supports are set to the ends of the girders to make the model stable (see *Figure 7*). The simulation consists of two steps (duration of each is 1 s): in the first step, only gravity is active. In the second step, both gravity and thermal load are activated. Supports are naturally active during both steps. Forces due to vacuum pumping could also be taken into account at this step of the analysis, but in our study the vacuum forces were left out of the analysis for now, but will be taken into account at a later testing stage.
Hint: For taking gravitation into account, it is better to use Standard Earth Gravity than Acceleration. Using Acceleration with downwards pointing component will cause the opposite effect than Standard Earth Gravity.

In the structural analysis, initial temperature is a crucial parameter because it sets the reference temperature for thermal deformations of the components. If initial temperature is set to 20 °C, this is the zero-deformation temperature, which can be the temperature where the component alignment is done. Then one can for example study the effect of only variating the ambient temperature, or as in our case, the effects of both the component heating and cooling as well as the ambient temperature change.

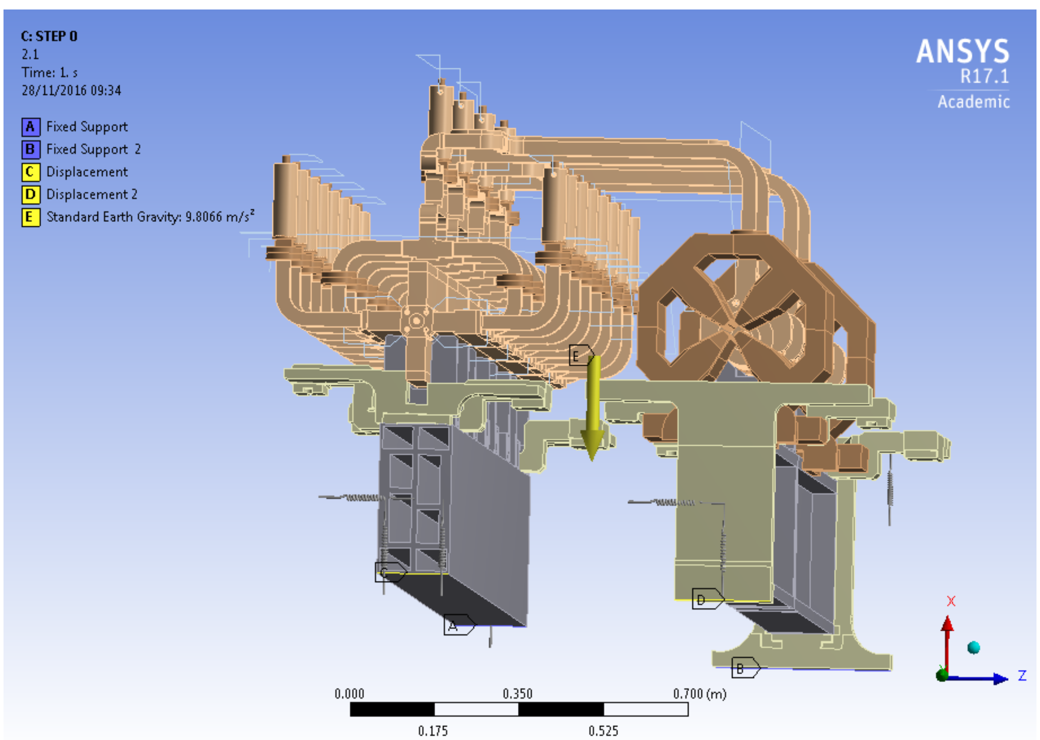


Figure 7. Definition of the mechanical supports and loads for structural analysis. Here (A) and (B) are fixed supports on edges at the end of the MB and DB girders, (C) and (D) are displacement supports on edges in the end of both girders with x-directional movement blocked, (E) is the standard earth gravity.

3.7 Test procedure

Thermal simulation follows the procedure of corresponding thermal tests (to be) conducted at the lab module type 0#2. The planned thermal test program consists of 6 operational cases described in *Table 3*. The cases vary by ambient temperature, reference temperature (important for deformations of the components), as well as heating and cooling. For example, in case 4 the ambient temperature is 40 °C and reference temperature for deformations is 20 °C with heaters and water cooling active. Case 5 is otherwise the same than case 2 but now the reference temperature is 40 °C. Thus, by comparing the results of these two cases one can distinguish the effect of component heating and cooling on the resulting deformations with respect to plain thermal expansions at 40 °C tunnel temperature.

Inlet water temperature is constant 25 °C and volumetric flows according to the CLIC baseline values [1]. The air speed is assumed to be constant 0.7 m/s, which yields to heat transfer coefficient of around 4 W/(m^2 °C) in all studied temperatures. Dropping the air speed to 0.4 m/s or below would change the theoretical heat transfer coefficient slightly to 2 or 3 W/(m^2 °C). In [5] it was noted that the influence of the air speed on the resulting temperature distribution is negligible. However, as mentioned earlier, heat transfer to air should be studied more thoroughly in the future. When comparing the results of the experimental tests to the simulation, the heat transfer coefficient for air flow could be used as a fitting parameter as was done in [5]. The steps of the thermal test program are summarized in *Table 3*.

Table 3. Steps of the thermal test program.

Case #	Input											
	T_∞ (°C)	T_{ref} (°C)	HTC (W/m^2 °C)	Heat power (%)			$T_{i,water}$ (°C)	Cooling water flow (m^3/h)				
				SAS	PETS	CL		SAS1	SAS2	SAS3	SAS4	PETS
0	20	20	4	100	100	100	25	0.069	0.069	0.069	0.069	0.037
1	40	20	4	0	0	0	25	0	0	0	0	0
2	40	20	4	0	0	0	25	0.069	0.069	0.069	0.069	0.037
3	40	40	4	0	0	0	25	0.069	0.069	0.069	0.069	0.037
4	40	20	4	100	100	100	25	0.069	0.069	0.069	0.069	0.037
5	40	40	4	100	100	100	25	0.069	0.069	0.069	0.069	0.037

3.8 Simulation results

From the results, we are mainly interested in the deformations occurring in the components of the module. For direct comparison with the corresponding experimental measurements, Deformation Probes are set for the positions of all Coordinate Systems created previously. Thus, we obtain the deformation of each fiducial point (or, to be precise, the closest point on a surface of the component) and are able to compare with the experimental measurements. Solution is obtained using ANSYS Remote Solver Manager (RSM) 17.1 with computational resources provided by CERN.

Hint: Since there are a lot of tracked points around the module, it is useful to export the results from the Deformation Probes as output parameters so they can be easily transferred to table format for post-processing and further analysis.

4. Summary and outlook

In this paper, the steps of thermal-structural Finite Element Analysis (FEA) of CLIC lab module type 0#2 were described. In geometry model simplification, some unnecessary components were first deleted or deactivated in CATIA (V5) to make the CAD model lighter. Then the geometry was imported in ANSYS (version 17.1) for suppressing more components and small details. Suppressing components and details enables using less elements in the analysis, which decreases the amount of required computational resources. Cooling water circuits were modelled by sketching lines and curves along the cooling water channels in and around the components. After model simplification and creating the cooling system, the geometry was imported in ANSYS Workbench for setting up the FEA.

The FEA consists of Steady-State Thermal and Static Structural analysis systems which were linked so that the results of the thermal analysis are imported as temperature distribution to the bodies in the structural analysis. In the structural analysis, also gravity was taken into account. The structural analysis consists of two simulation steps, where gravity is activated already in the first one, whereas the thermal load only in the second step. The planned thermal test program consists of 6 different cases with varying ambient and reference temperatures as well as the activation of the component heating elements and cooling water circulation.

In lab module type 0#2, the positions of the components are measured from fiducial points by a laser tracker. To be able to perform direct comparison with measurements, deformations are tracked in FEA by placing Coordinate Systems and assigning Deformation Probes for the points corresponding to the fiducials. The deformations are retrieved from the simulation as output parameters so they can be imported to tabular form.

There is always space for improvements in the simulation. In the future, heat convection by air flow should be more realistically taken into account in the simulation. Currently, a constant heat transfer coefficient is assumed for all the outer surfaces of the components. In reality, the air flow around the module varies and thus also the heat transfer coefficient varies. One approach would be to accompany the current thermal-structural analysis with another analysis system for the air using ANSYS Fluent. The different analysis systems can then be linked or coupled in ANSYS Workbench. Another improvement would be to define the contacts between components better. Now all the contacts in the model are bonded contacts with the same stiffness value. However, in the real lab module there are also more flexible contacts as well as sliding ones involved.

Next the FEM simulation is evaluated by comparison with the experimental thermal-structural measurements. The finite element model presented in this paper serves as a tool for studying the effect of different parameters and geometries for future module designs.

References

[1] S. Stapnes *et al.*, 2012, A Multi-TeV linear collider based on CLIC technology: CLIC Conceptual Design Report. CERN 2012-007.

[2] R. Nousiainen *et al.*, 2010, Studies on the Thermo-Mechanical behaviour of the CLIC Two-Beam Module. Linear Accelerator Conference 2010, Tsukuba, Japan.

[3] R. Raatikainen, 2011, Modelling of the thermo-mechanical behavior of the two-beam module for the compact linear collider, Master's thesis, Department of Mechanics and Design, Tampere University of Technology, Finland.

[4] F. Rossi *et al.*, 2012, Study of the Thermo-Mechanical Behaviour of the CLIC Two-Beam Modules. Proceedings of IPAC2013, Shanghai, China.

[5] L. Kortelainen, 2013, Thermo-mechanical modelling and experimental validation of CLIC prototype module type 0, Master's thesis, Department of Mechanical Engineering, University of Oulu, Finland.

[6] A. Xydou *et al.*, 2014, Thermo-mechanical tests for the CLIC two-beam module study, Proceedings of IPAC2014, Dresden, Germany.

[7] S. Griffet *et al.*, 2012, Strategy and validation of fiducialisation for the pre-alignment of CLIC components. The International Particle Accelerator Conference, May 2012, New Orleans, USA.

[8] T. Harris, 10 Useful New Features in ANSYS Mechanical 16.0, available in <http://www.padtinc.com/blog/the-focus/10-useful-new-features-in-ansys-mechanical-16-0> (Nov. 11th, 2016)

[9] <http://www.engineeringtoolbox.com/dry-air-properties-d_973.html> (Nov. 21th, 2016)

[10] F. Rossi, 2013, Thermal test program for CLIC prototype module type 0, EDMS 1320625, CERN, Geneva.