

Temperature mapping methods for thermoelastic analyses of the ARIEL spacecraft payload module

Andrés García-Pérez^{a,*}, Alejandro Fernández-Soler^a, Gianluca Morgante^b, Javier Pérez-Álvarez^a, Gustavo Alonso^a, Laura García-Moreno^a, Antonio Scippa^c, Daniele Gottini^c, Riccardo Lilli^c

^a Instituto Universitario de Microgravedad “Ignacio Da Riva” (IDR), Escuela Técnica Superior de Ingeniería Aeronáutica y del Espacio (ETSIAE), Universidad Politécnica de Madrid (UPM), Pza. Cardenal Cisneros 3, 28040 Madrid, Spain

^b Istituto Nazionale di Astrofisica (INAF), Osservatorio di Astrofisica e Scienza dello Spazio (OAS), via Piero Gobetti 93/3, 40129 Bologna, Italy

^c Dipartimento di Ingegneria Industriale, Università degli Studi di Firenze (UniFI), via St Marta 3, I-50139 Florence, Italy

ARTICLE INFO

Keywords:

ARIEL
Prescribed average temperature
STOP analysis
Temperature mapping
Thermal analysis
Thermoelastic analysis

ABSTRACT

The ARIEL mission is a European space project that aims to detect exoplanets with a spacecraft orbiting around the L2 point of the Sun-Earth system. The main payload consists of a Cassegrain telescope composed of mirrors that reflect and concentrate the incoming light from the deep space observations to finally guide it to the detectors. As in many other space missions, a dedicated complex assessment is established during the design phase to evaluate the impact on the optical performance caused by thermoelastic effects, which involves the coordinated work of the thermal, structural, and optical engineers. Despite that well-known and standardized processes and tools are established separately in each involved area, there is a lack of standardization about the way of exchanging the data between them, where additional calculations are required in some cases. This work focuses on the temperature mapping, which is the intermediate step between thermal and structural analyses, where temperatures are transferred to the structural model. The main difficulty of this process is related to the differences in modelling methods and approaches between both models, being necessary the development of an adequate algorithm to find the most accurate transfer of temperatures. This paper shows two different options for temperature mapping, detailing the proposed flowcharts. One of these methods requires the performance of an additional thermal conductive analysis, where a new improved procedure has been implemented in this work to solve some computational issues that made its application for large models difficult or even unfeasible. Both temperature mapping methods have been applied to the payload module of the ARIEL spacecraft, comparing the output results in terms of temperatures, stresses, forces, and displacements to evaluate their differences.

1. Introduction

During space missions, spacecraft payloads are exposed to harsh thermal environments, caused by the combination of their operations in vacuum, the heat transfer by radiation from the different celestial bodies and the variability of the internal temperature generated by the change of the incoming external heat fluxes during the different orbital phases. In addition, the different operational modes of the internal equipment (e.g. nominal, survival, idle, etc) lead to a variation of internal heat flows. Therefore, a crucial aspect during the design of spacecraft and their equipment is the evaluation of thermoelastic effects on the structure and especially on the payloads caused by the variability of the

internal temperature. Space telescopes are one of the different types of payloads on board spacecraft, which are present in scientific space missions for the distant observation of the stars, galaxies, and other celestial bodies [1–5]. The requirements for this type of payloads are very stringent with regard to the dimensional stability, since small changes in the mirrors or in their mechanical assembly in terms of displacements, angular distortions or deformations will cause a significant degradation in the optical performance.

One of the main causes that may lead to the degradation of the dimensional stability of space systems is the change in temperature along the mission [6]. To evaluate the effects on the optical performance of the thermoelastic phenomena [7,8], a design process denominated

* Corresponding author.

E-mail address: andres.garcia.perez@upm.es (A. García-Pérez).

<https://doi.org/10.1016/j.actaastro.2024.07.009>

Received 22 March 2024; Received in revised form 6 May 2024; Accepted 3 July 2024

Available online 6 July 2024

0094-5765/© 2024 The Authors. Published by Elsevier Ltd on behalf of IAA. This is an open access article under the CC BY license (<http://creativecommons.org/licenses/by/4.0/>).

Structural, Thermal and Optical Performance (STOP) analysis is established for space projects during the development phase of these systems. This process involves a complex coordinated work between three different engineering disciplines, which usually belong to different organizations. Therefore, it is recommended to define from the beginning of the project a careful interrelation of the common aspects and the procedure for a better efficiency, focusing on the way of transferring data between the different areas. Generally, the direct path in the STOP analysis process is defined in sequential order. The first step is to perform thermal analyses using as input the different conditions of each relevant thermal load case expected during the orbital phase to calculate the temperature map on the analysed spacecraft. A subsequent step called temperature mapping consists of transferring these temperatures to the structural model, which are used as input for the structural thermoelastic analyses to compute the displacements, rotations, and deformations on the different elements of interest. Finally, these displacement fields are transferred to the optical team to evaluate whether the degradation on the optical performance is within the allowed limits or not. This general process has been implemented in several space projects, each one with some particularities and small differences. In Ref. [9], the STOP analysis is defined for LISA space mission, where together with the accurate calculation of the thermoelastic effects on this spacecraft, the gravitational effects are also added to the process, renamed as STOP-G, considering its relevancy for this project dedicated to study the gravitational waves. The same need for accurate numerical simulations of the STOP analysis is presented in Ref. [10] for the design of future National Aeronautics and Space Administration (NASA) missions for planet finder spacecraft. In Ref. [11], the STOP process was adopted to investigate the effects of various design modifications on the developed interferometer performance to improve its design in the future. In Ref. [12], the application of the STOP analysis to the James Webb Space Telescope program is explained, showing the correlation of the predicted numerical analysis with the experimental results. The experimental verification of the numerical results calculated by the STOP analysis in a small CubeSat is detailed in Ref. [13]. The STOP analysis is also adopted for bigger ground-based telescopes [14], where similar software tools are used for the different main steps, but evaluating different thermomechanical environments according to the on ground conditions.

The three main tasks of the STOP process (thermal, structural and optical analyses) are individually well established in the space industry by the definition of general rules and guidelines [15,16] and by using their corresponding commercial software (ESATAN-TMS, SINDA and Thermal Desktop for thermal analysis, NASTRAN, ANSYS and ABAQUS for structural analysis and Zemax and CODE-V for optical analysis). However, there is a lack of standard procedures and commercial software for the common data exchange between each involved area. The first transition step corresponds to the temperature mapping between thermal and structural models. The main difficulty is the definition of the data transfer methodology, since both the thermal and structural models may have been built based on different numerical methods (lumped parameters or finite elements), with different modelling assumptions (detailed vs simplified) and with different mesh sizes.

Four temperature mapping methods are exposed in Refs. [7,8], which have also been tested in other previous works [17,18]. These methods mainly focus on the transmission of temperatures between a thermal model created by the lumped parameter method, being ESATAN-TMS the most used software in the European space industry, and a structural model defined by the finite elements (FE) approach, usually in NASTRAN code.

The simplest temperature mapping method is the so-called Patch-Wise Temperature (PWT) method, which consists of the direct transfer of the temperature calculated in each thermal node to the set of structural nodes associated to it. This association is usually done by the nearest-node technique, which finds for each structural node the nearest thermal node. This approach is extensively used in several projects [1,

19]. Other more sophisticated search algorithms involve the calculation of the shapes and sizes of both thermal and structural entities to establish with more accuracy the correct association between the geometries of the structural and the thermal models. Typically, the thermal model mesh is much coarser than the structural FEM mesh. Therefore, there are several structural nodes associated to the same thermal node. This method has the advantage of its simplicity, since only geometric computations are performed to define the groups of structural nodes or elements associated to each thermal node. The transferred temperature map is visualized on the FEM as a set of zones or patches, each one with a constant temperature and geometrically similar to each corresponding thermal node. The drawback of this method is that there are temperature jumps between the patches that are not realistic and, in some cases, lead to conservative results in terms of local stresses or forces.

The second temperature mapping option is the Geometrical Interpolation (GI) method, which is an approach that attempts to provide a smoother temperature distribution on the FEM, avoiding the patches of constant temperature generated from the previous method. This second option firstly transfers the temperature of each thermal node to a single structural node, typically the closest one, and then calculates the temperature field for the rest of structural nodes by employing an interpolation algorithm, which only considers the geometric parameters such as the distances between the nodes. This option is used in some works related to the STOP analysis process [5,20]. The main drawback is the strong dependency of the resulting temperature maps with the selected interpolation algorithm and its corresponding input parameters, which may lead to inaccurate maps if these input parameters are established a priori arbitrarily.

The third method is called the Centre-Point Prescribed Temperature (CPPT), which represents a more advanced approach in which the temperature maps on the structural finite element model (FEM) are calculated in additional thermal analyses with the same FEM, but with properties converted from structural to thermal. The thermal FEM analysis uses as boundary conditions the temperatures transferred from each node of the lumped parameter model (LPM) to its closest FEM node, like in the previous method. In this case, the conversion of the FEM from structural to thermal properties requires an important effort involving the close collaboration between the thermal and the structural engineers, being this aspect the main drawback of this method. This FEM conversion mainly consists of the properties modification by setting the conductive parameters of the materials and joining elements while maintaining the same nodes and most of the elements. The purpose is to refine the temperature distribution on the FEM by performing linear thermal conductive analyses. However, if this obstacle is overcome, in principle the results are more reliable than those from the previous methods because the temperature distributions have been computed by assuming the thermal equilibrium in these additional thermal conductive analyses. This assumption is more realistic than considering constant temperature patches as in the first method, or an arbitrary geometrical interpolation as in the second one. Some works about the STOP analyses of different space structures have used this method to transfer temperature maps from the thermal analyses to the FEM [12, 21].

The fourth and most accurate method described in Ref. [8] is the Prescribed Average Temperature (PAT) method, which is an improved version of the previous one. The PAT method is based on two assumptions.

- The temperature calculated at each thermal node of a lumped parameter model corresponds to the average temperature of the zone represented by this node.
- The temperature map can be refined with the FEM assuming that it is in a quasi-steady-state equilibrium considering only the conductive properties of the structure (without any thermal radiative exchange).

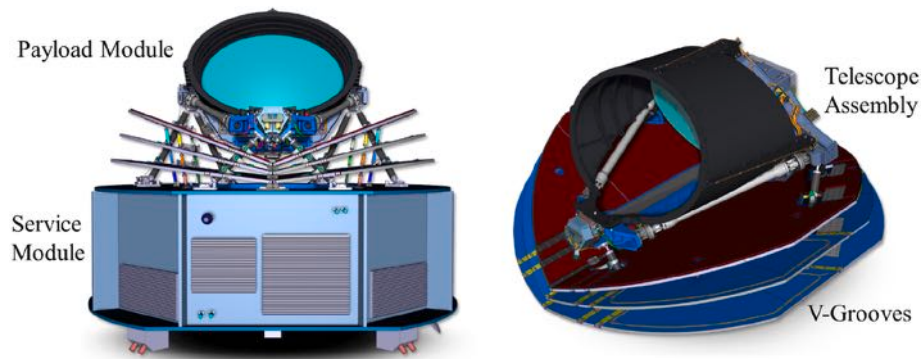


Fig. 1. CAD of the ARIEL spacecraft (left) and its Payload Module (right).

Taking into account the above-mentioned assumptions, the PAT method described in this work uses a thermal conductive analysis with the FEM to calculate the final temperature map, similar to the CPPT method. However, the CPPT method presents the inaccuracy of assuming that the temperature calculated at each LPM node represents the local value at the centre point of this node. Actually, the calculated value represents the average temperature of the entire region covered by the node. The PAT method considers this aspect by implementing an additional condition to guarantee that the average temperature on each FEM region associated to a single LPM node corresponds to the temperature transferred from this LPM node. Therefore, the PAT method is the option of the four methods described in Ref. [8] that provides the most realistic temperature fields on the FEM. The PAT method has been implemented in a temperature mapping software tool called SINAS [22], which has been developed over the last decades and is currently being improved by ESA in an updated tool called pysinas [23]. This software tool has been designed to perform the temperature mapping between a thermal LPM built in ESATAN-TMS and a structural FEM defined in NASTRAN. For the geometric association between both models, the software tool considers not only the position of the nodes, but also the geometric shapes of the elemental entities. This process requires complex calculations to evaluate the sizes and shapes of both the thermal nodes and the structural elements in a step denominated “overlapping” of models. For the old SINAS IV software, the overlapping process success strongly depended on the similarity between the modelling approaches adopted for both models. Therefore, it was usually required that both the thermal and the structural models were represented by elements of the same dimensionality, mostly 2D elements, which limited the versatility of this tool for models with different modelling approaches and mismatches in geometric definitions. For instance, structural models are continuously growing in quantity of nodes due to the increasing use of 3D elements to obtain more accurate results, especially required for the STOP analysis in several projects, while the thermal models are still composed of 2D geometries. This is the handicap of the analysed structure described in this paper, whose structural FEM is mostly composed of 3D elements while the geometry of its thermal model is defined by surfaces. The current pysinas software has solved this limitation, being now versatile with any combination of geometries between LPM and FEM.

One of the purposes of this work is the development and application of a new software tool for temperature mapping, which have been successfully used for the Atmospheric Remote-Sensing Infrared Exoplanet Large-survey (ARIEL) mission. This project is being carried out by the European Space Agency (ESA) with the objective of observing exoplanets while orbiting in a halo orbit around the L2 Lagrange point of the Sun-Earth system [24,25]. The spacecraft is composed of two main modules, the Payload Module (PLM) consisting of a Cassegrain Telescope that will operate at cryogenic temperatures around 50 K and the Service Module (SVM) containing the auxiliary subsystems such as

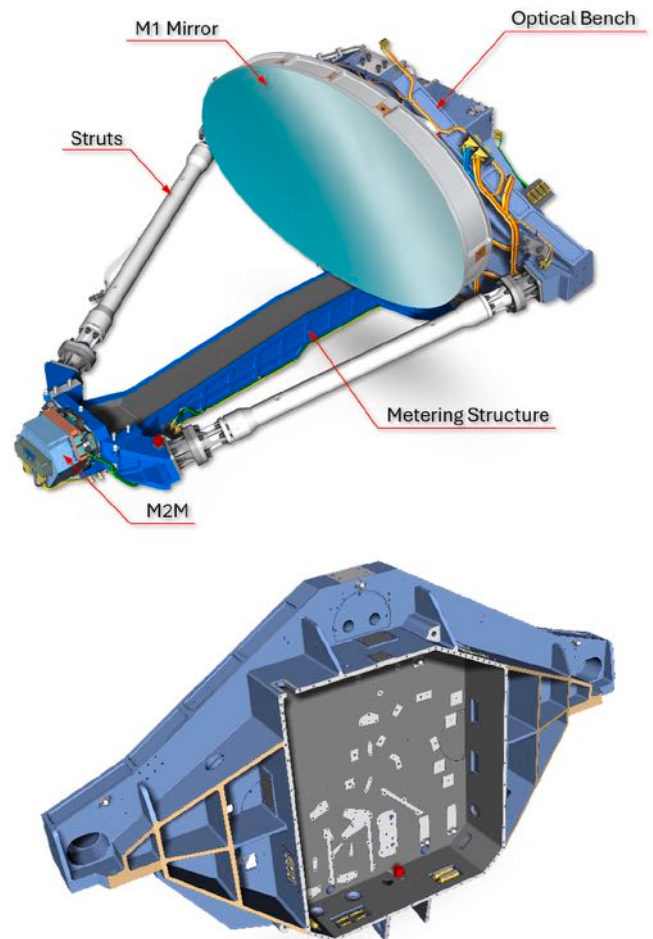


Fig. 2. Cad of the ARIEL telescope assembly (top) and optical bench (bottom).

avionics and the communication equipment (see Fig. 1).

The main challenge of the Telescope Assembly (TA) of the ARIEL PLM (see Fig. 2) is the design of a full aluminium structure that will undergo a significant thermoelastic contraction from the room temperature to the operational cryogenic environment due to the relatively high coefficient of thermal expansion (CTE) [26]. This material was selected based on its lower price and better thermal conductivity that minimizes the temperature gradients during observation. The preliminary analysis from the optical point of view about these effects is depicted in Ref. [27]. It shows the great complexity of considering the different sources of misalignment degrading the optical performance

during operation. One of the main parts of the TA structure is the Optical Bench (OB), an aluminium part whose complex geometric design is the result of a careful process involving the thermal and structural numerical models [28] to achieve a final design compatible with a full set of requirements. Most of the requirements applied to the OB correspond to the mechanical, thermal and electrical interfaces with most of the equipment on board the telescope assembly, including the main mirror M1. The M1 primary mirror is a monolithic stiffened part made of aluminium supported by three flexure hinges. Its design is also challenging [29] due to the combination of its large size (major axis of 1.1 m and minor axis of 0.7 m) with the high CTE value of the aluminium material, which leads to a significant thermoelastic contraction that should be with the minimum angular and shape distortion. Different tests were conducted on representative samples made of the same aluminium material [30,31] to find the best way to perform the heat treatment that achieves the mechanical stabilization of the primary mirror. A detailed FEM of the primary mirror was initially created to study the behaviour of this critical part under gravity environment in Ref. [32], where numerical analyses were key in discovering the causes of some failures during prototype tests, saving the costs associated with the execution of additional tests.

The thermal architecture of the ARIEL PLM is described in Refs. [33, 34], where the stringent thermoelastic stabilization is achieved by a combination of passive and active cooling elements. Passive temperature control is provided by three V-grooves and one radiator that allow the telescope assembly to be in a stable temperature below 60 K. Active cooling control (Neon Joule-Thomson cryocooler [35]) is added to decrease the temperature below than 42 K because the detectors of the ARIEL Infra-Red Spectrometer (AIRS) [36] must operate at such low temperature during the operational observation periods. The numerical calculation of the temperature on the entire ARIEL PLM is done by a thermal model built in ESATAN-TMS. The temperature maps of different representative scenarios are transferred to the structural FEM of the same assembly. Then, the resulting displacements and deformations of the optical elements are evaluated through the corresponding thermoelastic analyses with the aim of assessing the impact on the optical performance. This STOP analysis process should be done with the maximum possible accuracy considering the sensitivity of the optical results. With this purpose, a careful assessment was done in Ref. [37] to evaluate the causes of the numerical inaccuracies in the structural thermoelastic analysis of the ARIEL PLM and define the way to improve the precision of the calculated displacements.

The work presented in this paper describes the procedures and application of two different temperature mapping methods (PWT and PAT) performed for the ARIEL PLM in the framework of the STOP analysis to compare the results and evaluate their differences. The procedures defined for both mapping methods are exposed in section 2, while the STOP process used for the ARIEL project is explained in section 3, together with the description of thermal (ESATAN-TMS) and structural (NASTRAN) models of the ARIEL PLM. For the PAT method, an additional thermal conductive analysis is required as an intermediate step to refine the temperature map on the structural FEM, where the structural properties of the FEM are replaced by thermal properties to perform this thermal conductive analysis. In order to assure in this analysis that the average temperature of each FEM region corresponds to the value of the associated ESATAN-TMS node, extra entities, such as multipoint constraints (MPC), are added in the FEM. However, some issues related to the required high computational resources have been found in these analyses caused by MPC entities connected to a great number of nodes, making the application of the PAT method for large models difficult or even unfeasible. Fortunately, this work proposes a solution consisting of dividing the original large MPCs into a set of smaller MPCs. The computational assessment described in section 4 demonstrates the validation of the proposed solution, where the computational resources are appreciably reduced while maintaining the same degree of accuracy in the results. Thanks to that, the PAT method

can be applicable to the ARIEL PLM FEM for temperature mapping. The comparison between the PWT and PAT temperature mapping methods in terms of temperature distributions on the ARIEL PLM FEM is shown in section 5, together with the structural output (stresses, forces and displacements) calculated by the subsequent thermoelastic analysis. Finally, the conclusions are summarized in section 6.

2. Methodology

This work focuses on the transfer of temperature between a thermal model built with the lumped parameter method and a structural model defined with the finite element method. The software widely used for thermal analysis in the European space industry is ESATAN-TMS, specifically designed by the European Space Agency (ESA) and supported by the company ITP Aero to calculate the temperature results on spacecraft where the main heat transfer mechanism is by radiation. On the other hand, the most employed structural analysis code in the space industry is NASTRAN, designed by the National Aeronautics and Space Administration (NASA), which remains popular due to its reliability for linear simulations.

The lumped parameter method is based on the discretisation of a continuous medium as a discrete network of nodes representing the capacitance of the system linked by conductive (G_L) and radiative (G_R) couplings. The basic justification for this spatial discretisation is the isothermal assumption, meaning that each node is considered as isothermal. Hence, in regions where spatial temperature gradient exists, more nodes are needed to fulfil the isothermal assumption. Supposing a spacecraft orbiting a planet, discretized as a model of N nodes, where C_i is the heat capacity, T_i is the temperature, $\dot{Q}_i$ is the internal heat load, and $\dot{Q}_{load,i} = \dot{Q}_{P,i} + \dot{Q}_{albedo,i} + \dot{Q}_{Si}$ is the environmental heat load absorbed by the node i , the energy balance is approached by Ref. [38]:

$$C_i \frac{dT_i}{dt} = \sum_{j \neq i}^N K_{ij} (T_j - T_i) + \sum_{j \neq i}^N R_{ij} (T_j^4 - T_i^4) + \dot{Q}_i + \dot{Q}_{load,i}, \quad (1)$$

where the linear couplings G_L between node i and node j is K_{ij} , and the radiative exchange constant between those nodes is R_{ij} . Therefore, solving the N first-order differential equations with enough boundary conditions, the temperature of the nodes and the heat fluxes exchange between them are obtained. This approach is implemented in the software ESATAN-TMS.

The finite element method is a numerical approach consisting of dividing a complex structure into several small elements with relatively simple geometric shapes to represent its behaviour by a discrete set of nodes. Unlike the lumped parameter method, each FEM node does not have any associated volume or area, but they represent point locations of the structure. FEM nodes are connected by elements, which are the entities with the associated dimensions. This is the most employed method for structural analysis, where the simple geometries of the elements allow the easy definition of the stiffness, damping and mass matrices for each element according to its corresponding formulation. These elemental matrices are assembled for the entire model considering the full set of degrees of freedom (displacements and rotations of the nodes), which are later used to solve the mechanical equations. For linear static analysis, only the stiffness matrix $[K]$ is used, together with the vector of applied loads $\{P\}$, to calculate the vector of resulting displacements $\{u\}$ by the following equation:

$$[K]\{u\} = \{P\} \quad (2)$$

The rest of required results (stresses, forces, etc.) are later computed according to the constitutive equations for each type of element.

Thermoelastic analysis is a particular option for linear static analysis that evaluates the mechanical effect on a structure caused by the temperature change. Therefore, the inputs are the initial or reference temperature map (T_{ref}) and final temperature map, which are initially used

to define a set of equivalent nodal forces P_i by Ref. [7]:

$$P_i = \sum_{j=1}^N E_j A_{ij} \alpha_j (T_{ij} - T_{ref}), \quad (3)$$

where N is the number of elements connected to the node i , E_j and α_j are the Young's modulus and the coefficient of thermal expansion respectively of the element j , A_{ij} is a geometric parameter that depends on the formulation of each type of element considering its shape and size, and T_{ij} and T_{ref} are the final and initial (or reference) temperatures respectively. Typically, the initial temperature map is constant and reflects the stress-free or undeformed temperature field, while the final temperature map required for a thermoelastic analysis has been derived by mapping the temperature field produced by the lumped parameter thermal model on the finite element model.

Two temperature mapping methods (PWT and PAT) have been selected in this work to carry out the temperature transfer from the thermal model to the structural model of the ARIEL PLM to evaluate the thermoelastic effects within the framework of the STOP analysis. The Patch-Wise Temperature (PWT) method is one of the most used temperature mapping methods in the space industry due to its simplicity, which consists of the direct definition of the temperature between each thermal node and the associated group of structural nodes. Its main drawback is that the temperature map generated on the structural FEM is composed of constant temperature patches (each patch corresponds to the influence zone of each thermal node), which is not completely realistic. A way to improve the accuracy of the mapping process is by using a more realistic approach, such as the Prescribed Average Temperature (PAT) method. This method provides more realistic temperature distributions on the FEM, but with the need of performing intermediate thermal analyses with the same FEM to compute these final temperature maps. The main efforts are focused on the conversion of the FEM, initially defined only for structural analyses, to be able to perform thermal conductive analyses. Additionally, a way to assure that the average temperature of each FEM region corresponds to the temperature transferred from the associated ESATAN-TMS node is necessary. This can be done by adding some extra entities in the thermal FEM such as for instance multipoint constraint (MPC) elements. These are the main difficulties of this method, especially the FEM conversion, which requires the teamwork between the thermal and structural engineers. Furthermore, it was found in this work that if MPC entities containing a great quantity of independent nodes, on the order of 10000 nodes, are employed for temperature averaging, high computational resources are required, which may make FEM thermal analysis difficult or even unfeasible. Due to the interesting advantages of the PAT method in terms of accuracy and realism, one of the objectives of this work is the assessment of this method for the ARIEL project and the development of a methodology to overcome its difficulties.

A software tool called “ESATAN2NASTRAN” has been developed by IDR/UPM institute with the objective of performing temperature maps between a thermal model defined in ESATAN-TMS and a structural model defined in NASTRAN. The tool was developed to provide temperature mapping in different methods, being the PWT and the PAT the used ones for the ARIEL PLM. The detailed definition of both mapping methods is explained below.

2.1. Patch-Wise temperature method

The Patch-Wise Temperature (PWT) method has been extensively used in several projects to perform in a relatively easy way the temperature transfer from a thermal model to the structural model. As it was above-mentioned, the main difficulty in temperature mapping is the establishment of a process to link the nodes or elements of the structural model to the corresponding nodes of the thermal model. This process can be called “association” between structural and thermal entities and typically involves geometric calculations. One option for this process is

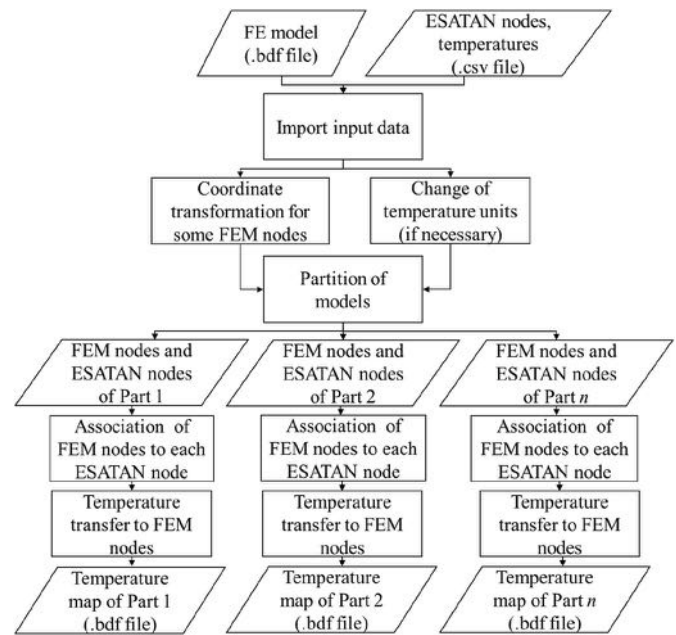


Fig. 3. Flowchart of the temperature mapping process with the Patch-Wise method for the ARIEL PLM.

used by SINAS IV software [22], denominated as “overlapping” of models, where the geometric shapes of the thermal and structural models are carefully considered to establish the classification of groups of structural elements that represent the same area as each corresponding thermal node. Additionally, some extra operations are needed to correctly classify the structural elements that are located on the edge of two adjacent thermal nodes. Therefore, the approach used by this software is the association of dimensional geometric entities instead of points, what requires the complete definition of the geometry of both models. One of its drawbacks is the difficulty of correctly establishing this association when there is a significant difference between both models in terms of geometric accuracy, representativity and type of mesh (2D mesh vs 3D mesh). This difference in geometric representation of the structure in both models can be a source of errors in the mapped temperature fields and hence a source of errors in the final thermoelastic responses.

For the temperature mapping of the ARIEL PLM, it was decided to use a simpler algorithm for the structural and thermal entities association. Firstly, the structural entities associated in this process are the nodes instead of the elements. Secondly, the search algorithm only considers coordinates referring to the same coordinate system for the nodes of both the thermal and structural models. The algorithm, based on the nearest-node approach, finds the closest thermal node for each structural node. Therefore, the shapes and sizes of the structural elements and thermal nodes are not considered, avoiding the problems related to the complexity of evaluating a great variety of types and shapes of elements, the possible mismatch between both models in some complex areas and the uncertainty related to the final association of the border elements. The only required data from the thermal model are the list of nodes indicating their identification (ID) numbers, coordinates, and temperatures for the analysed load case. An extra field is included to indicate the name of the part or zone to which each thermal node belongs, named label, which can be useful for the model partition explained below. All these data are compiled in a simple.csv file, which is imported in the mapping software tool. In a similar way, the data required from the structural model are the list of nodes indicating their ID numbers and positions. The temperature mapping software can read all this data directly in NASTRAN format (.bdf file), which can also include other data such as elements, properties, and coordinate systems, which in

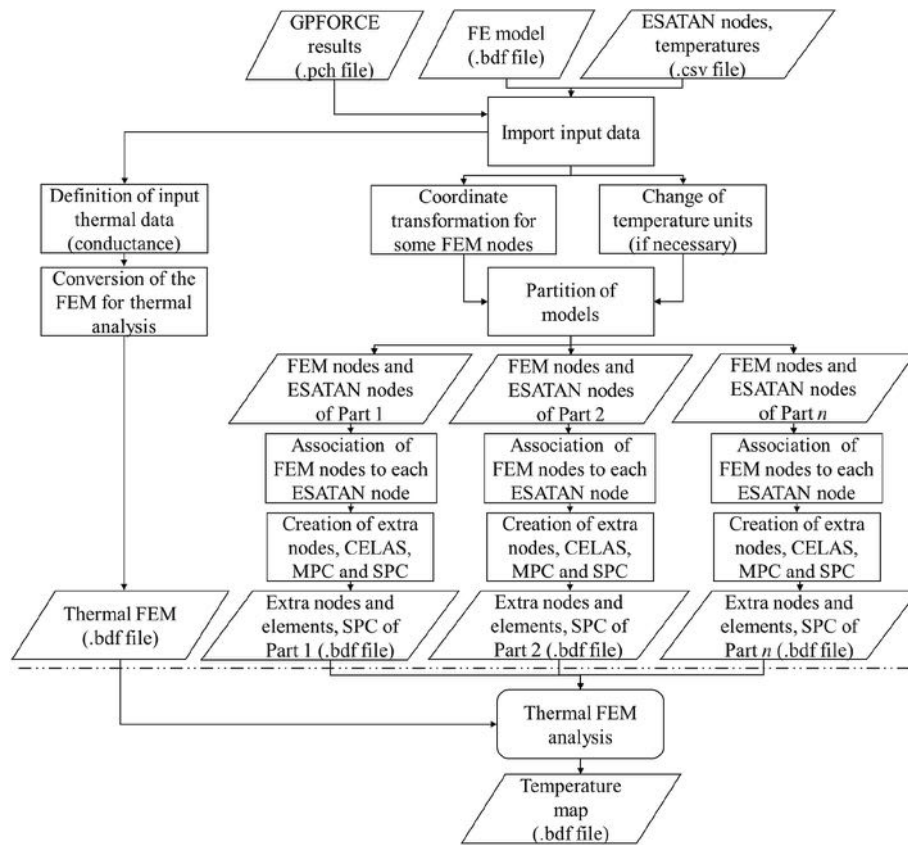


Fig. 4. Flowchart of the temperature mapping process with the PAT method for the ARIEL Payload.

principle are not mandatory for the search algorithm, but they can help in classifying the structural nodes according to their represented part or subsystem for the model partition process explained below. Each resulting patch may be very similar to the corresponding ESATAN-TMS node when the thermal mesh is regular (elements with the same shape and size), but it may be noticeable different for complex zones with irregular mesh. However, this drawback is compensated by the versatility of this approach, compatible with any combination of different types of elements used to model the analysed system.

The flowchart designed for the PWT mapping in the developed software is depicted in Fig. 3. The input data correspond to two files, a csv file with the required data from the thermal model, and a bdf file with the definition of the structural FEM in NASTRAN format. Both files are read to import all the necessary information from both models. In case that the position of some of the FEM nodes are referred to a different coordinate frame than the one used by the thermal model, a coordinate transformation process of these nodes is needed at the beginning. The mapping software has the possibility of reading the entries that define the coordinate frames from the bdf file and perform the corresponding coordinate transformation. To do this, it is necessary that all the entries of the affected coordinate frames are defined in the bdf file, including at least one reference frame coincident in orientation and location with the one used for the positions of the thermal nodes. This coincident coordinate system is selected by the user for the coordinate transformation step.

On the other hand, the temperatures read from the csv file may be in different unit system than the one required for the subsequent FEM analysis. Therefore, the software offers the possibility of selecting both the imported and the final desired units for the temperatures to perform this conversion before the mapping execution. Furthermore, a constant value of temperature can be defined to be added to all the temperatures to change the origin of the temperature scale if necessary. This can be

useful for the subsequent thermoelastic analysis when the input final temperatures need to be defined as deltas of temperatures (difference between the initial temperature and the final temperature) instead of the corresponding absolute values.

The next step in the flowchart is the division of the analysed structure into different parts or subsystems applied for both the thermal and structural models. This process is identified as “Partition” of models and is a key step for the quality and the execution time of the mapping procedure. The objective is to perform a division in the maximum possible number of parts, the greater the number of divisions the better, according to the representation of each part in both models and the easiness of selecting the groups of nodes of each part. These parts are typically identified based on functional representation in both models. In many cases this approach also simplifies the selection of nodes in both models into groups. Initially, the user should have a sufficient knowledge about both models to identify the definition of each part in these models. Depending on the level of detail employed in the representation of the same part in each model, the modelling approach, and the geometrical simplifications, a part can be anything from a single piece to a complex subsystem.

The main and largest structural parts of the ARIEL PLM, such as the Optical Bench, Metering Structure and the principal mirror M1 among others, are separately selected in this partition process as single pieces. On the other hand, smaller complex subsystems, such as the M2 mirror mechanism, have all their internal components selected for the same group due to the difficulty of individually selecting each internal component for the thermal and the structural models. The objectives of the partition step are the following.

- Avoiding the unintended association between a structural node belonging to a subsystem and a thermal node of another different subsystem.

- Reduction of the execution time during the subsequent search step. This is because to find the closest thermal node for each structural node, the search involves only the thermal nodes of the corresponding part rather than all the thermal nodes of the entire model.

The mapping software tool provides two ways of performing the definition of the set of nodes for each part for both the thermal and the structural models. The first option is by indicating the numbering ranges of the nodes, while the second option is by selecting the names of the zones or regions belonging to each division. In case of the structural FEM, the regions are identified by the names of element properties, while for the thermal model an extra field in the imported csv file contains the name of the region associated to each node.

After partitioning both models into different groups, the software executes the temperature transfer to each structural node from the corresponding closest thermal node. This process is done part-by-part, that is, the search algorithm involves only the thermal nodes of the analysed part in that moment, reducing the execution time compared to a process without this partition. In any case, this step is the most time-consuming, where the distance between each FEM node and each thermal node is calculated and compared to finally find the closest thermal node. Finally, this closest thermal node is associated to the structural node that is being evaluated. It is expected that each thermal node is associated to several structural nodes due to the finer mesh resolution of the FEM compared to the thermal model. For the Patch-Wise Temperature method, the next step is the direct temperature transfer from each thermal node to all the structural nodes associated to it. Consequently, this generates a temperature map on the FEM composed of patches, hence the name of this method, each one with a constant temperature that may be different from the temperatures of the adjacent patches. Once all the structural nodes of a part have their corresponding temperature, the algorithm performs the same search algorithm for the next part.

Finally, the output data are the nodal temperature for each FEM node written by the NASTRAN entry “TEMP”. These results are written in different bdf files, each one related to the corresponding part, which will be used as input for the subsequent structural thermoelastic analysis.

2.2. PAT method

The implementation of the PAT method is significantly more complicated than the previous PWT method. Essentially, the PAT scheme (shown in Fig. 4) is an ampliation of the PWT scheme, where apart from the objective of reading the temperature from the thermal model to be transferred to the FEM, it is necessary to include a second path to convert the FEM from structural to thermal simulations. Therefore, extra activities are added to the flowchart to perform this more sophisticated procedure.

Some of the steps described for the PWT method are still used for the PAT method, in particular, the importing of both models, the operations of selecting the desired units for temperatures, coordinate transformation of some of the FEM nodes and partition of both models remain the same. One different aspect is related to the imported FEM. While for the PWT method it was not necessary to include in the input bdf file some typical entries such as the materials, for the PAT method, where the same FEM will be converted for thermal simulations, it is necessary to include all the relevant entries for the complete definition of this model. Another different aspect is the objective of the main path. While for the PWT it was the creation of the output files containing the temperatures for all the FEM nodes, for the PAT method, this path finishes with the creation of the boundary conditions (fixed temperatures defined by SPC entries for some extra nodes), together with extra nodes, conductive elements (CELAS) and multipoint constraints (MPC) that will allow the calculation of the temperature map in a subsequent thermal conductive analysis.

The objective of the main path is the same association between each

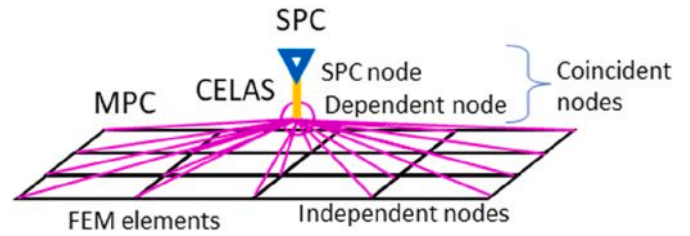


Fig. 5. Scheme of the extra entities (centre nodes, MPC, CELAS, SPC) added to the FEM for the PAT mapping method.

FEM node with its closest thermal node, also defined for the PWT method. However, this association is used for the PAT method to create the aforementioned extra entities in the FEM to guarantee that the temperature for each thermal PLM node is transferred to the FEM. However, instead of directly applying the temperature to the associated set of FEM nodes, a multipoint constraint (MPC) is created for each zone influenced by a thermal node connecting all the FEM nodes of this zone with an extra node (see Fig. 5). By default, this extra node is located with the same coordinates as the thermal node (in reality, the position of this extra node will not affect the calculated results). This extra node is the dependent node of the MPC, allowing the temperature of this node to be the average temperature of all the independent nodes connected to this MPC. The problem is the incompatibility of directly setting the temperature of this dependent node by a single point constraint (SPC). To overcome this inconvenient, an extra duplicate node is added (by default in the same position as the dependent node) with the fixed input temperature (by the SPC), which will be transferred to the dependent node in the analysis by an extra conductive element (CELAS) with a very high conductive value (for the ARIEL PLM simulations it was set to 1.0×10^5 W/K). In this way, both extra nodes will have the same input temperature during the FEM thermal analysis, which corresponds to the temperature of the related thermal PLM node.

The MPC establishes a linear relationship between the temperature of the dependent node, T_d , and the temperatures of the N independent nodes, T_i by the following equation:

$$w_d T_d = \sum_{i=1}^N w_i T_i \quad (4)$$

For the definition of the MPC, it is necessary to configure the weighting factor for each connected node, with the condition that the sum of all the weighting factors for the independent nodes, w_i , is equal to the absolute value of the weighting factor of the dependent node, w_d , to correctly perform the averaging.

$$\sum_{i=1}^N w_i = w_d \quad (5)$$

In Ref [7], it is commented that one of the most realistic ways to set these weighting factors is by considering the associated volume for each node. There are different options to calculate this parameter, but the used one for this work is by assuming that the influence zone of each PLM node corresponds to a portion, part or group of parts made of the same material. If this assumption is correct, the associated volume for each FEM node is proportional to its associated mass, which in turn is proportional to the resulting nodal force when the structure is subjected to a uniform gravity load in a static analysis. Therefore, it is necessary to previously run this static gravity load analysis with the same FEM to request the nodal forces (by the NASTRAN command “GPFORCE”) and import these results (by a.pch file) into the software tool to automatically configure the weighting factor (w_i) of each FEM node by the next equation:

Table 1

List of NASTRAN entries modified for thermal analysis.

Category	Structural entries	Thermal entries	Description
Materials	MAT1	MAT4	Entry for isotropic material definition
	MAT8	MAT5	Entry for orthotropic and anisotropic material definition
Element properties	PCOMP	PSHELL	Entry for composite 2D property
	PBARL, PBEAML, PBUSH	PBAR, PBEAM, PELAS	Entries for 1D bar and beam properties
			Entry for property of the joining 1D elements
			Entry for the joining 1D elements
Elements	CBUSH	CELAS	Entries for multipoint constraints
Multipoint constraints	RBE2, RBE3, RBAR	MPC or CELAS	Entries for multipoint constraints

$$w_i = \frac{\left| \vec{F}_i \right|}{\sum_{j=1}^N \left| \vec{F}_j \right|} \quad (6)$$

Where $\left| \vec{F}_i \right|$ is the module of the resulting applied force to the node i .

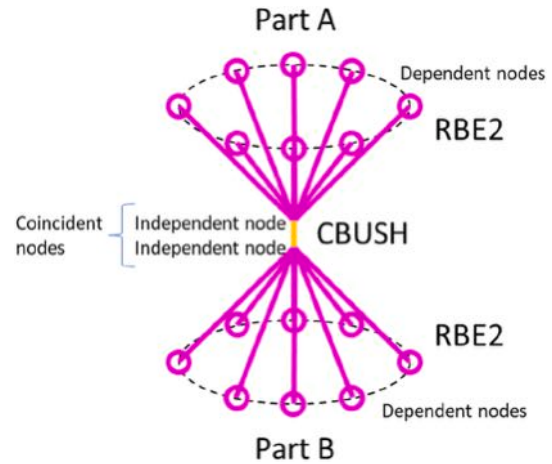
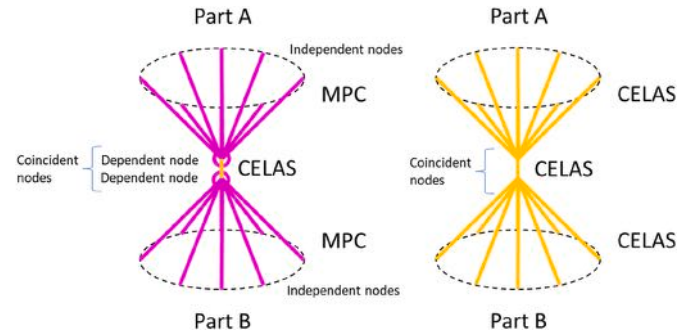
With this equation, the sum of weighting factors of all the independent nodes associated to the same MPC (N nodes) will give a unitary value.

The end step of this main path is the creation of output files containing the definition of the boundary conditions (fixed temperatures by SPC entries), together with the extra nodes, CELAS elements and MPC entries, which are used in a subsequent thermal analysis of the FEM.

The second path is a new procedure added for the PAT method with the aim of preparing the FEM for thermal analysis, where most of its entities remain the same (nodes, coordinate systems, 2D and 3D elements), but there are other entries (materials, joint elements, rigid elements) that need to be tailored with thermal parameters, mainly thermal conductivities. The NASTRAN entries that shall be modified are indicated in Table 1, where some particular aspects are clarified in the explanation below.

Material definitions used for structural analysis, with parameters such as the Young's Modulus and Poisson's Coefficient, among others, should be replaced by the material entries with thermal parameters such as the thermal conductivity. Isotropic structural materials (MAT1) are replaced by equivalent isotropic thermal materials (MAT4), while other more complex materials definitions (orthotropic MAT8) are replaced by equivalent definitions for orthotropic and anisotropic thermal materials (MAT5). PCOMP entries are used to define composite materials, as well as for 2D sandwich properties. This entry seems not compatible with thermal analysis. Therefore, it should be substituted by an equivalent monolithic PSHELL property whose thickness is the total thickness of the original PCOMP property. Additionally, new MAT5 entries are created in this process to define the equivalent thermal materials associated to these new shell entries, where the thermal conductivity in each direction is carefully calculated according to the replaced composite or sandwich structure.

On the other hand, two options are available in NASTRAN to define the cross-section parameters for the 1D elements: by defining the section shape and its dimensions, or by directly provide its associated geometry parameters such as the area, second moments of area and shear and torsional modules. It seems that the first option is not compatible with thermal analysis in NASTRAN versions up to 2021, so a conversion of the affected property entries (PBARL and PBEAML) to the equivalent entries of the second option (PBAR and PBEAM) is necessary. For the thermal analysis, only the cross-sectional area is needed for the conductance of 1D elements, which is calculated in this conversion considering the different possible shapes and dimensions.


Fig. 6. Typical scheme for the representation of the mechanical joints in a structural FEM.

Fig. 7. 2 options for FEM joining entities conversion: MPC approach (left) and CELAS approach (right).

The key aspect in this FEM conversion is the definition of the joining entities, where the modelling of the corresponding thermal conductivity should be carefully performed. Modelling of the joints in the structural FEM is typically done by the combination of rigid elements (mostly RBE2) and elastic 1D elements (CBUSH). The scheme for modelling these joints is depicted in Fig. 6, where the CBUSH elements should be of zero length to avoid numerical errors for the thermoelastic analysis.

For the conversion of this set of entities, two options are available, which are shown in Fig. 7. In both options, the original CBUSH element is replaced by a CELAS element, which is the NASTRAN entry used for generic 1D conductive elements for thermal analysis. The only needed parameter of these conductive elements is the value of the thermal conductance, which is indicated in the associated property entry (PELAS). For the first option, each RBE2 is replaced by a thermal multipoint constraint (MPC), where the centre node is the dependent one, whose temperature will be the average of the temperatures of the independent nodes. By default in this case, all the independent nodes have the same weighting factor, where the total sum of these factors is 1. The second option consists of replacing each original rigid element (RBE2) by a set of new CELAS elements, each of them connecting the centre node to each external node.

The conductance values used for these joining elements have been extracted from the ESATAN-TMS model in their respective interfaces, where the thermal conductance associated to a screw is the total conductance between the two connected parts divided by the number of screws between both parts. For the option 1 (MPC approach), this resulting thermal conductance is directly defined for the central CELAS element. For the option 2 (CELAS approach), a more complex calculation is done, considering that there are 3 sets of conductive elements in

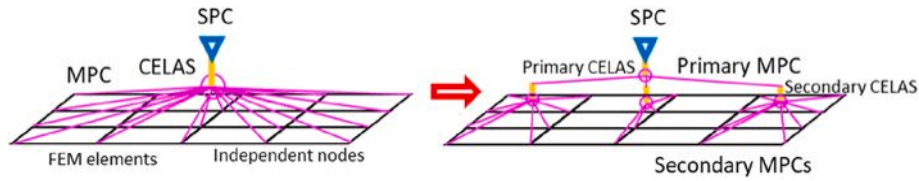


Fig. 8. Scheme for the modification of MPC entities for PAT method: original MPC (left) and new set of MPCs and CELAS elements (right).

series (from replaced RBE2 (part A), CBUSH and RBE2 (part B)) and assuming that the CELAS elements that replace each original rigid element are quasi-parallel. Taking both assumptions into account, and deciding by default that the thermal conductance is equally distributed between the 3 sets of conductive elements, the resulting conductance values are calculated by the following equations:

$$GL_i = 3GL_T \quad (7)$$

$$GL_{ij} = \frac{GL_i}{N_i} \quad (8)$$

where GL_T is the total conductance of the represented joint and GL_i is the conductance of each set of conductive elements. GL_i is directly the value of the thermal conductance for the central CELAS element. Additionally, GL_i is also the resulting thermal conductance of each set of N_i conductive elements that replace a single rigid element, considering that all these CELAS elements are almost parallel and have the same individual thermal conductance of GL_{ij} .

Once all the thermal parameters are configured for the FEM, the final step of this path is the generation of a NASTRAN file (bdf) with the converted FEM definition, which is used later as input file, together with the other files with the additional entities, in the corresponding thermal analysis to refine the temperature map on its nodes. The results of this analysis correspond with the temperature calculated to each structural node by the PAT method, which are used as input in the subsequent structural analysis to calculate the required thermoelastic results.

One of the issues detected when performing the FEM thermal analysis in NASTRAN versions up to 2021 is the exigent computational resources required for the simulations in terms of RAM memory and hard disk storage. Some rehearsal analyses were made with a simple model, and it was concluded that these computational issues were caused by the calculation of the dependency relationships between the nodes belonging to the MPC entities, in particular for those MPCs with the greatest quantity of independent nodes. A new method to modify these MPCs without altering the results was introduced into the procedure, where those MPC entities with more independent nodes than a given value are divided according to the scheme shown in Fig. 8.

In the example shown in Fig. 8, the original MPC has 20 independent nodes, and it is decided that the maximum allowable number of independent nodes is 8, for instance. Therefore, this original MPC is replaced by a set of MPCs defined in 2 levels, where the lower level MPCs (secondary MPCs), directly connected to the structure, have a maximum of 8 independent nodes (in the example there are 2 MPCs with 8 independent nodes each one, and an additional MPC with the remaining 4 nodes). These secondary MPCs are joined via secondary CELAS elements to one upper level MPC (primary MPC), which is in turn connected via the original primary CELAS element to the node with the fixed temperature (SPC). In principle, secondary CELAS do not seem to be necessary since the dependent nodes of the secondary MPCs can be at the same time independent nodes of the primary MPC. However, it was checked that the configuration without secondary CELAS elements requires the same high computational resources as the original one. Therefore, the addition of the secondary CELAS is fundamental to achieve the reduction in the computational resources. Furthermore, they should have a very high thermal conductance. The weighting factors of the independent nodes of the secondary MPCs remain the same as in the original MPC. To

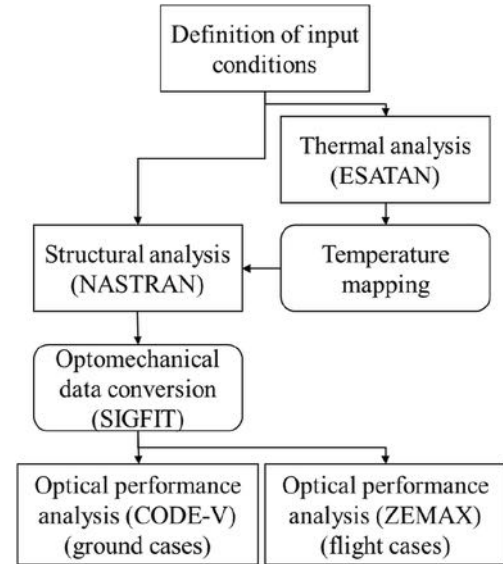


Fig. 9. Flowchart of the STOP analysis for ARIEL Payload.

maintain the same average temperature for the corresponding FEM zone as that provided by the original MPC, the absolute value of the weighting factor for the centre node of each secondary MPC is the sum of the original weighting factors of its independent nodes. Furthermore, this resulting weighting factor should be the same for the independent node of the primary MPC connected via a CELAS element to this centre node of the secondary MPC. This approach is also compatible for schemes with 3 or more levels of MPCs.

This approach is validated in section 4 of this paper with a case study with a simple FEM of a square plate modelled by 2D quadrilateral elements to evaluate the influence of the maximum number of MPC independent nodes on the computational resources required to perform the thermal analysis. Additionally, the same assessment is done with the ARIEL PLM FEM to confirm the applicability of this method in a generic complex structure.

3. STOP analysis in ARIEL mission

The Structural, Thermal and Optical Performance (STOP) analysis is a critical assessment for the success of the ARIEL mission. The impact of different thermoelastic and gravity environments on the ARIEL telescope optical performance is evaluated. A set of different load cases is established to cover the different key scenarios, which are classified into two groups: on-ground and in-flight load cases. The STOP analysis flowchart for the ARIEL Payload is shown in Fig. 9. The first step is the definition of the input conditions for each load case for both thermal and structural analyses when applicable. The boundary conditions that define the thermal analysis cases are typically the radiative environment, the power mode of the instruments and/or by setting temperatures and heat fluxes as boundary condition at particular thermal nodes. Then, to perform a thermal analysis, firstly, the Geometrical Mathematical Model (GMM) is built to calculate the thermal radiative

exchanges between the different thermal nodes and the incoming heat fluxes from the thermal environment in each radiative case. Secondly, the Thermal Mathematical Model (TMM) that includes all the conductive couplings, the power dissipation at each thermal case and the radiative couplings obtained from the radiative case is generated. Then, the thermal analysis can be run to obtain the conductive and radiative heat flows and the temperature of each thermal node.

The temperature map from each thermal analysis load case is used as input for the subsequent structural analysis. Before the corresponding structural analysis, it is necessary to transform the temperature map from the thermal model to the structural model, considering the difficulties derived from the different methodologies employed for each type of analysis (thermal analysis is based on lumped parameter method while structural analysis is based on the finite element approach) and the differences in the average sizes of the elements between both models. This key aspect is evaluated in the next sections of this paper.

To calculate the deformation in the main elements of the Telescope Assembly, the temperature maps from the thermal analyses, the definition of the gravity vector orientations and boundary conditions (mostly with enforced motions at the IF nodes) are used as input conditions for the structural analysis in each load case. Additionally, stresses and forces are calculated for the structural evaluation in some of the load cases. The displacements and rotations of the nodes that represent the mirrors and other secondary optical elements are exported to the optical teams. In this project there are two different teams of optical engineers, one dedicated to the evaluation of the ground cases and the second one dedicated to the flight cases. For both teams, data exportation is done in the same way, by using an intermediate software (SIGFIT) [39] that receives as input the displacement fields of the analysed mirrors to generate the optical data that are used as input for the optical analyses. Finally, each optical team evaluates these data to calculate the final performance parameters that determines whether the system is compliant or not with the established requirements.

In this work, three load cases of the ARIEL STOP analysis have been used for the assessment of the temperature mapping method. The first case corresponds to the “Operational Cold” load case, which is a steady-state condition where the Telescope Assembly is at a nearly uniform cryogenic temperature slightly above 50 K. It has been selected to be a representative case for the optical performance. The second case is the “Non-Operational Cold” load case, which is another steady-state condition, but with lower quasi-uniform temperature. Therefore, being the coldest steady-state scenario for the ARIEL mission, it is one of the worst thermoelastic conditions from a stress point of view. The third case is the “Max Gradient” load case, where after a transient thermal analysis simulating the flight cool-down during the journey to L2 in the transfer orbit [40] (similar to Planck [2] and Euclid [41] missions), the temperature map corresponding to the instant with the maximum gradient between the M1 mirror and the OB is used for the subsequent structural analysis. Unlike the other two cases, this load case shows appreciable temperature gradients between the parts, which will cause high shear forces on the joint elements. The main issue related to these shear forces is the possible slippage between the connected parts, which may have a non-negligible negative effect on the optical performance due to micromotions of the optical elements with regard to their theoretical positions.

3.1. Thermal and structural models for ARIEL PLM

As above mentioned, the Payload Module of the ARIEL spacecraft is designed to ensure that both the instruments and the telescope operate at temperatures below 60 K. This fact is achieved through a combination of passive control elements: three V-grooves, the instrument radiator, and by using insulating struts that mechanically attach the PLM to the SVM. The SVM, which houses all the electronics, operates in the range between 253 and 323 K. The TA consists of four mirrors, the M1 baffle, which blocks unwanted light rays from reaching the M1, and the

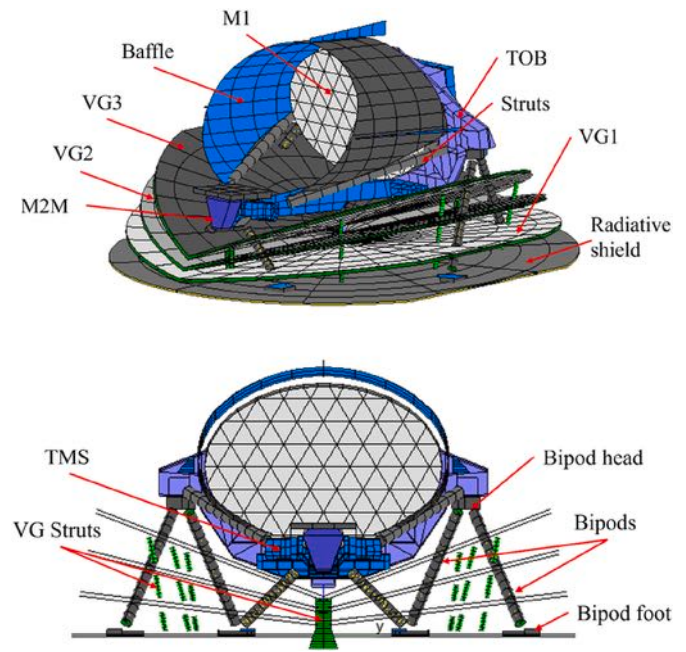


Fig. 10. Overview of the ARIEL GMM.

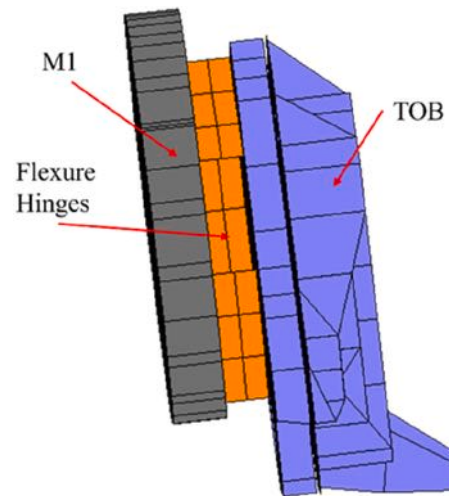


Fig. 11. M1, flexure hinges and OB GMM.

telescope structure, which consists of the OB, the metering structure (TMS) and the struts. In addition, the telescope structure is black painted, achieving that the baffle and the OB, which are very large, act as radiating surfaces to enhance the PLM thermal performance.

As previously commented, the ARIEL thermal model has been built in ESATAN-TMS. The GMM and TMM is based on the CAD model and represents the main units [34] to verify the thermal requirements. The GMM consists of shell geometries as it is shown in Fig. 10, most of them with quadrilateral shape. The whole ARIEL thermal model consists of 4905 thermal nodes.

As above mentioned, M1 is mechanically attached to OB via three Flexure Hinges. Due to its surface finish (low IR emissivity), M1 is highly radiatively decoupled from its surrounding elements, and therefore, its thermal behaviour is driven by OB through its I/F with the Flexure Hinges. The GMM of the M1-Flexure Hinges-OB is shown in Fig. 11.

In the ARIEL project, the use of two different finite elements (FE) models for the structural analysis at PLM level was decided. The first one is the detailed PLM FEM used mainly for the structural simulations for

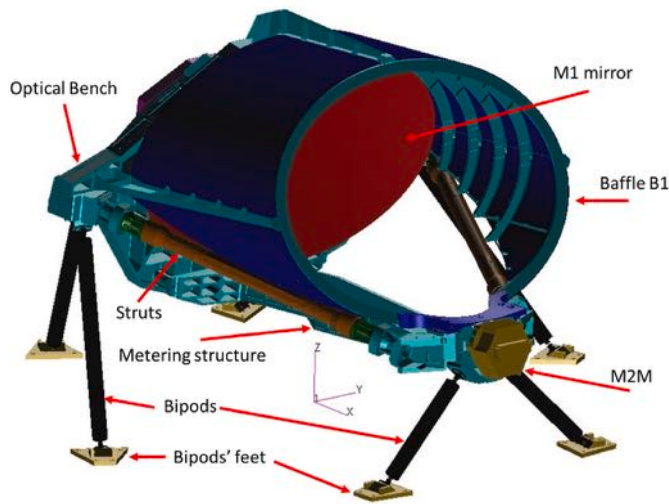


Fig. 12. Detailed FEM for ARIEL PLM.

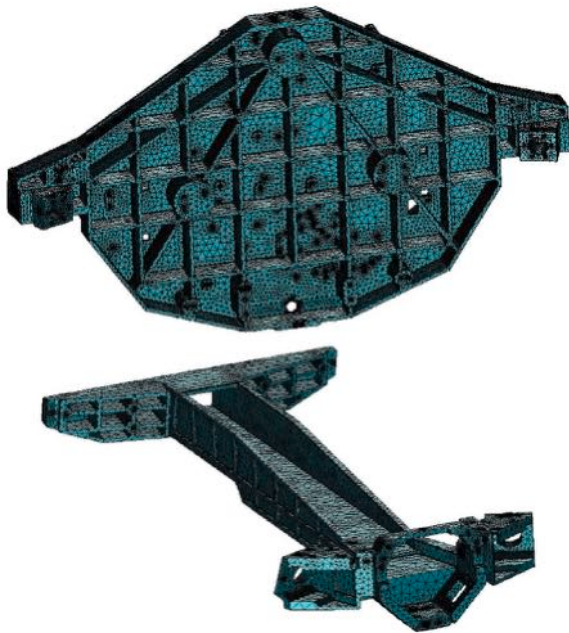


Fig. 13. Detailed FE models for the Optical Bench (top) and Metering Structure (bottom).

the STOP analysis, considering the need for high accuracy in terms of structural results for a better optical performance evaluation during the subsequent optical analysis. In addition, the structural simulations of STOP analysis are linear and static, which allows the use of a detailed FEM with several nodes and elements due to the lower computation requirements in terms of execution time and memory storage. On the other hand, the second model is used for the structural simulations with dynamic mechanical environments such as sine vibration or acoustic. It consists of a FEM created mostly by the 2D elements approach to represent the main structural parts of the entire ARIEL PLM. Consequently, this second FEM is considered as the simplified PLM FEM, which is more suitable for dynamic analyses where the computation requirements are much more severe than for static analyses. Furthermore, this simplified FEM is still adequate for stresses and forces calculation taking into account that the degree of accuracy is less stringent compared to the STOP analysis results. Therefore, the FEM used in this work is the detailed one, whose characteristics are indicated

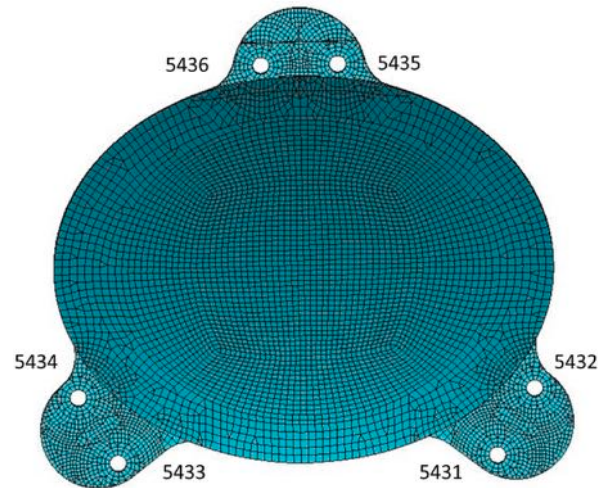


Fig. 14. FE model for the secondary M2 mirror with the ID numbers of the joint elements (CBUSH).

below.

The detailed ARIEL PLM FEM used for the STOP analysis has been used during the different design phases, increasing the level of detail in the geometric representation by using mostly 3D elements to represent the main structural parts of the Telescope Assembly (TA), including the mirrors. The main structure of the TA was modelled by the team of the Spanish research institute Ignacio Da Riva (IDR), which belongs to the Universidad Politécnica Madrid (UPM). Additionally, the detailed FE models of different subsystems created and delivered by the corresponding responsible teams have been assembled into the detailed PLM FEM: the FE models of the M1 mirror with its flexure hinges and the M2 mirror have been provided by the Instituto Italiano di Astrofisica (INAF), the FEM of the mechanism of M2 mirror (M2M) was provided by the Spanish company SENER, the FEM of the baffle B1 was provided by the Portuguese company Active Space and the FEM of the bipods was provided by the Technical University of Denmark (DTU). Other subsystems also considered as part of the PLM, such as the V-grooves, have not been included in this PLM FEM because their mechanical contribution is negligible, since they are not directly mechanically linked to the Telescope Assembly. The version of the FEM used for the Cycle C1 (before the Critical Design Review) of the STOP analysis is shown in Fig. 12 with the PLM global coordinate frame, while the FE models of the Optical Bench and Metering Structure are shown in Fig. 13.

This version of the PLM FEM is composed of more than two million nodes, where the predominant type of elements is the tetrahedral with 10 nodes. The principal mirror M1 and its supports are mostly modelled by linear hexahedral elements, while thin parts such as the Baffle B1 and the mechanism that supports the secondary M2 mirror (M2M) are modelled by 2D quadrilateral elements. The joints between the parts are represented by infinitely rigid elements (RBE2) and 1D elastic elements of CBUSH type whose stiffness values can be adjusted for each interface, following the scheme shown in Fig. 6. Some of the results of interest for this project correspond to the interface forces on the joint elements (CBUSH). These forces are then used in a dedicated bolt analysis to guarantee that all the bolts have positive design margins, being especially critical the margins related to the slippage between the parts. This work will show the calculation of the interface (IF) forces between the secondary M2 mirror and its mechanism (M2M) caused by thermoelastic effects, where the ID numbers of the CBUSH elements are depicted in Fig. 14.

For each mirror, there is an interpolation element (RBE3) that connects all the nodes of the reflective surface to a single dependant node located close to its centre. Its main objective is the computation of the rigid body motion of this reflective surface in terms of translational and

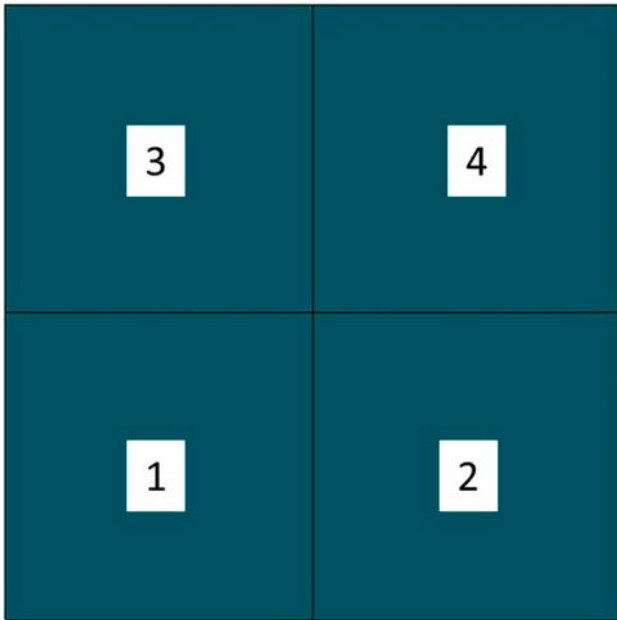


Fig. 15. Square plate with the identification of the four zones used for the computational assessment of thermal analysis.

rotational displacements. These results are part of the inputs for the subsequent optical performance analysis to measure the impact of the different environments analysed in the STOP analysis.

4. Computational assessment of FEM thermal analysis in NASTRAN

In this section, an assessment is carried out to evaluate the impact on the computational resources required for FEM thermal analysis caused by the presence of MPCs with high number of independent nodes. In NASTRAN, there are two solution sequences available for the steady-state thermal conductive analysis, the linear static SOL101 and the general-case steady-state thermal analysis SOL153, which takes into account nonlinear features for radiation and convection. In this work, the first option was selected due to its simplicity and faster simulations.

Two computational problems arise when performing a linear thermal analysis in NASTRAN (versions up to 2021) with a FEM model that contains MPC entities with a high number of independent nodes. The first one is related to the large amount of disk storage required for a temporal file (scratch file), which may exceed the maximum computer capacity. The second problem is related to the use of the RAM memory. When running a NASTRAN analysis, the used RAM memory is distributed into two main partitions, the Buffer Pool and the Open Core. It has been observed that the required memory for the second partition (Open Core) increases when analysing a FEM with large MPC entities. If the allocated memory for the Open Core partition is not enough for it, a fatal error message appears in the output f06 file indicating that there is insufficient memory, showing the estimated extra memory necessary to successfully finish the analysis.

To overcome these problems, a new method explained at the end of section 2.2 has been developed in this work, which consists of the MPC modification where each original MPC is substituted by a two-level set of smaller MPCs. This method is validated in this section with two different models. The first part of this assessment (subsection 4.1) is done in a simple representative FEM consisting of a square plate composed of one million quadrilateral elements and slightly more than one million nodes. The second part of this assessment (subsection 4.2) is done with the ARIEL PLM FEM described in section 3.1, but converted for thermal analyses with the PAT method, where the corresponding extra MPC

Table 2

Computational results of the square plate FEM thermal analyses.

Secondary MPCs		Primary MPCs		Required Open Core memory [GB]	Max hard disk usage [GB]	Success
Number of nodes	Quantity	Number of nodes	Quantity			
2000	500	125	4	15.0	39.4 ^a	No
1500	668	167	4	11.2	47.0	Yes
1000	1000	250	4	7.5	32.5	Yes
500	2000	500	4	3.8	22.2	Yes

^a This analysis was not successfully completed, and this value does not reflect the maximum expected hard disk usage.

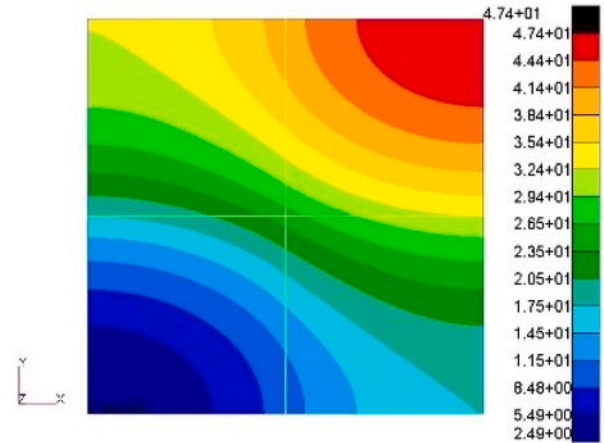


Fig. 16. Temperature map in °C with the PAT method for the square plate FEM – MPCs limited to 500 nodes.

entities have been included.

4.1. Computational assessment of FEM thermal analyses with square plate FEM

A representative FEM has been created for this case study, consisting of a square plate of 1 m side and 10 mm of thickness made of aluminium ($K = 167 \text{ W/(m K)}$) and modelled by one million 2D quadrilateral elements. The plate is divided into four zones as shown in Fig. 15, where each zone has a different known average temperature. That is, it is simulated that there exists a lumped parameter thermal model with 4 elements for the square plate and it is desired to transfer the temperatures from this thermal model to the FEM by the temperature mapping PAT method. In this particular case, the average temperatures are 10 °C for the zone 1, 20 °C for the zone 2, 30 °C for the zone 3 and 40 °C for the zone 4. Each zone has approximately 250000 nodes, which are initially connected to a single MPC. Therefore, four large MPCs of approximately 250000 independent nodes each are included in the FEM to average the temperature for each region during the thermal conductive analysis in NASTRAN.

After the computational problems found in the initial configuration caused by the presence of these four large MPCs, the proposed method of dividing them into smaller sets of MPCs was applied to overcome this issue. Different options were analysed by selecting different values for the maximum number of independent nodes for the secondary MPCs. When this number decreases, the quantity of secondary MPCs increases and vice-versa. The computational results of these analysed options are shown in Table 2.

As can be seen in Table 2, the analysed options use MPCs with significantly much less independent nodes than the original ones, which range from 500 to 2000 nodes. The maximum RAM memory assigned to

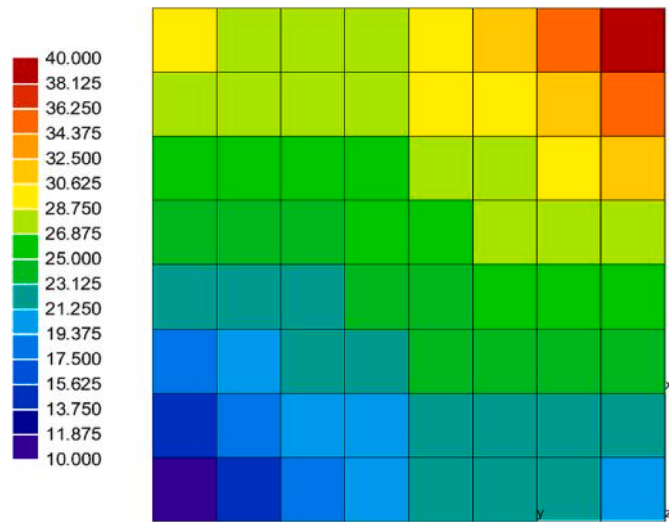


Fig. 17. Temperature in °C for an 8x8 thermal model of the square plate.

these analyses was around 12 GB. Therefore, the option with secondary MPCs of 2000 independent nodes each was not successfully executed. However, when the maximum number of independent nodes is reduced, the computational resources are reduced as well, allowing the execution of these analyses. It can be observed that the relationship between the required RAM memory for the Open Core partition and the number of independent nodes of the secondary MPCs is almost proportional. The resulting temperature map calculated by the FEM thermal analysis (PAT method) for the configuration of MPCs with 500 independent nodes is shown in Fig. 16. The distributions of temperature are practically the same for all options, with slight differences (less than 0.26 %) between them. It can be noticed that the range of the calculated nodal temperatures is wider than the range of the input average temperatures, giving the coldest temperature of 2.5 °C in the bottom-left corner and the hottest temperature of 47.4 °C in the top-right corner. This is because the set of input temperatures indicates the average value in each quadrant, while in the FEM the calculated temperature distribution with gradients is reproduced, providing values above and below the corresponding average temperature.

The PAT method proposed in this work presents some differences in the procedure compared to the PAT method described in Ref. [23] and implemented in the pysinas software, which is currently being developed by ESA. While in the proposed method the FEM nodes are directly associated to the corresponding thermal nodes using the nearest-node search algorithm, the pysinas software uses a more sophisticated

process to associate the FEM elements considering the shape geometry of each thermal node. Another difference is the way to compute the weighting factors for the averaging. While in the approach proposed in this work is based on associated masses (through gravity load analysis), in pysinas is based on associated volumes. The process to calculate the temperature is also a bit different between both options. For the pysinas software, the thermal conductive matrix together with the constraints matrix are first assembled, and then the final distribution of temperature in the FEM is calculated by solving the matrix system of linear equations. When trying to perform the temperature mapping of this square plate by the pysinas software, it was found that the process was not successfully completed due to computational issues, possibly caused by the physical memory limitation due to the in-core solver used by pysinas. In order to compare the results between the proposed method and the pysinas software, a new thermal model of the square plate with 8x8 thermal nodes was created, whose temperatures can be seen in Fig. 17. The resulting temperature maps from both options were very similar, showing relative errors below 0.26 %, as can be appreciated in Fig. 18. This great similarity in the resulting temperature maps for this simple case demonstrates that despite the differences in the procedure between the PAT method proposed in this paper and the one implemented in the pysinas software, they can give the same results in regions with uniform thermal nodes. However, the differences between both options increase when using a thermal model with non-uniform mesh because the more sophisticated overlapping module of the pysinas software takes into account the geometry shape of each thermal node, unlike the nearest-node search algorithm implemented in the procedure explained in this paper.

4.2. Computational assessment of the thermal analyses of ARIEL PLM FEM

The approach proposed in this work consisting of dividing the original large MPCs into a set of smaller MPCs has also been tested in the ARIEL PLM model (described in section 3.1) to validate its application

Table 3

Classification of the MPCs according to the number of their nodes for the original ARIEL PLM thermal FEM.

Number of independent nodes	Quantity of MPCs
<300	470
300–500	224
500–1000	350
1000–1500	126
1500–2000	70
>2000	215

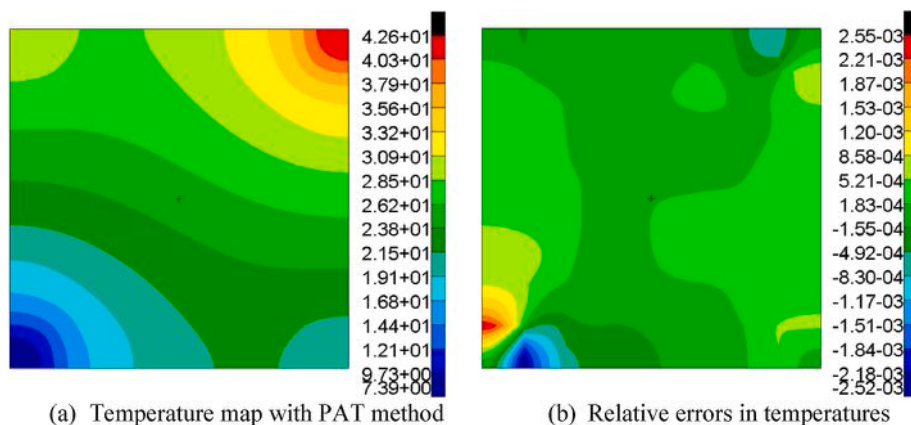


Fig. 18. Temperature map in °C with the PAT method for the square plate FEM (a) and relative error in temperatures between the PAT method described in this paper and the pysinas software (b).

Table 4
Computational results of the ARIEL PLM FEM thermal analyses.

Maximum number of nodes per MPC	Quantity of MPCs	Required Open Core memory [GB]	Max hard disk usage [GB]	Execution time [min]	Success
81574	1455	73.5	332.1 ^a	76 ^a	No
2000	2248	19.6	62.2 ^a	8 ^a	No
1500	2592	15.6	50.8 ^a	7 ^a	No
1000	3297	11.3	72.3	25	Yes
500	5572	6.3	52.4	20	Yes
300	8426	4.0	40.6	15	Yes

^a These analyses were not successfully completed. Therefore, the marked values do not reflect the maximum expected computation parameters.

for a complex FEM representative of a real structure. After the application of the PAT method described in section 2.2 to the ARIEL PLM, the original thermal FEM has a great variety of sizes for its MPCs, ranging from a few nodes to tens of thousands of them, generated according to

the differences between the amounts FEM nodes associated to their corresponding thermal PLM nodes. There are 1455 MPCs, which are classified according to the number of independent nodes in Table 3. The largest single MPC has 81574 nodes.

Due to the large sizes of some of the MPCs, the thermal analysis in NASTRAN of this original FEM exceeds the computational limits of the computer. Therefore, the process to transform the original MPCs into a tree configuration of smaller MPCs is applied to this FEM to evaluate the impact on the computational resources required by the corresponding thermal analysis. Different configurations are analysed, each one with a defined value for the maximum allowable number of independent nodes for the MPCs. Consequently, only those MPCs with more nodes than the corresponding limit are replaced by smaller MPCs in the two-level tree configuration explained at the end of section 2.2. The analysed configurations and their corresponding computational results are indicated in Table 4.

The first analysis corresponds to the original FEM, which after 76 min of execution stopped before finishing because the RAM memory required by the Open Core partition was much higher than the available

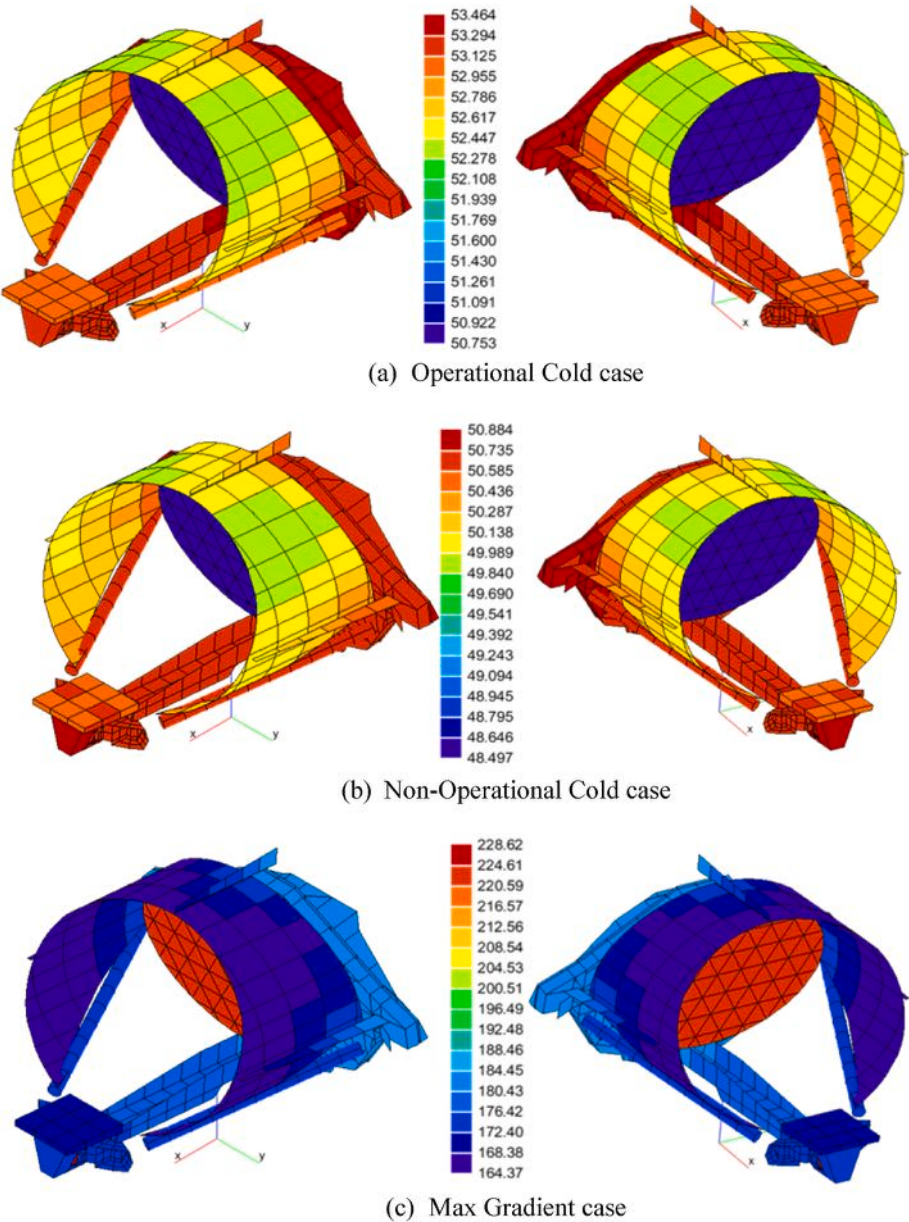


Fig. 19. Temperature maps (in Kelvin) obtained from ESATAN-TMS model on the Telescope Assembly.

one, which was about 12 GB for the used computer. The options with 2000 and 1500 for the maximum number of independent nodes also overpassed the computational limits in the same aspect. The RAM memory required for the Open Core partition can be known for those non-successful analyses thanks to the estimation of this required memory indicated in the generated output files that appears when one analysis fails. However, there is not any estimation about the maximum hard disk usage required for the non-successful analyses. The values marked with an asterisk in Table 4 are the computational results for the corresponding failed analyses, but they are not representative of the required maximum usage for the corresponding successful runs. On the other hand, when the limit for the maximum number of nodes for the MPCs was established equal or below 1000, the thermal conductive analysis of the FEM could be successfully completed. As can be observed, when reducing this limit, the quantity of MPCs increases due to the greater number of affected original MPCs that are converted into smaller MPCs and the greater number of these new smaller MPCs generated in this conversion. Reducing the limit on the maximum number of nodes is computationally beneficial because both the required RAM memory and hard disk usage decrease quasi-proportionally. Finally, it was checked that the output results in terms of temperature maps maintain the same degree of accuracy between the different options, showing maximum errors about 2 mK between the different successful analyses.

The conclusion of this assessment, done for both a simple square plate FEM and the complex ARIEL PLM FEM, is that reducing the maximum number of independent nodes of the MPCs is beneficial from computational point of view, allowing the execution of the thermal conductive analysis in NASTRAN in any FEM without loss of accuracy. This fact facilitates the employment of the PAT method for the temperature mapping for models with millions of nodes such as the ARIEL PLM FEM.

5. Numerical results of the thermal and structural analysis of the ARIEL PLM

This section describes the results calculated by the ESATAN-TMS thermal model and the NASTRAN structural model for the ARIEL PLM in the STOP analysis. Once the way to overcome the computational drawbacks of thermal analyses for the PAT method in FE models with many nodes has been determined in the previous section, the objective of this section is to show the comparison of the temperature map and the subsequent thermoelastic results (displacements, stresses and forces) between the PWT and the PAT methods to determine the differences between both methods in a real structure.

Three representative load cases from the STOP analysis of the ARIEL project have been assessed to show in this work the results and the comparison between the two studied temperature mapping methods: The Operational Cold case, as the representative scenario for the optical performance, and the Non-Operational Cold case and the Max Gradient case (maximum gradient between M1 mirror and Optical Bench during the cool-down), which are the worst cases regarding stresses and forces. The cold operational and non-operational steady state cases correspond to the nominal orbit at the Earth-Sun L2 point [34]. In both steady-state cases, the SVM (PLM IF) thermal nodes are defined as boundary conditions at 253 K. In addition, the instrument detectors and CFEEs are switched on in the cold operational steady-state case and switched off in the non-operational steady-state case. The Max Gradient transient case refers to the journey from the Earth parking orbit to the final orbit at the Earth-Sun L2 point. During the journey, all PLM units cool down to their operational temperature. Therefore, as thermal gradients may occur at the IFs of different units, the effect on mechanical performance must be assessed.

The final temperatures for each load cases have been calculated with the ESATAN-TMS model described in section 3.1 and the corresponding input conditions. The temperature maps on the Telescope Assembly for the three assessed load cases are shown in Fig. 19, while the same

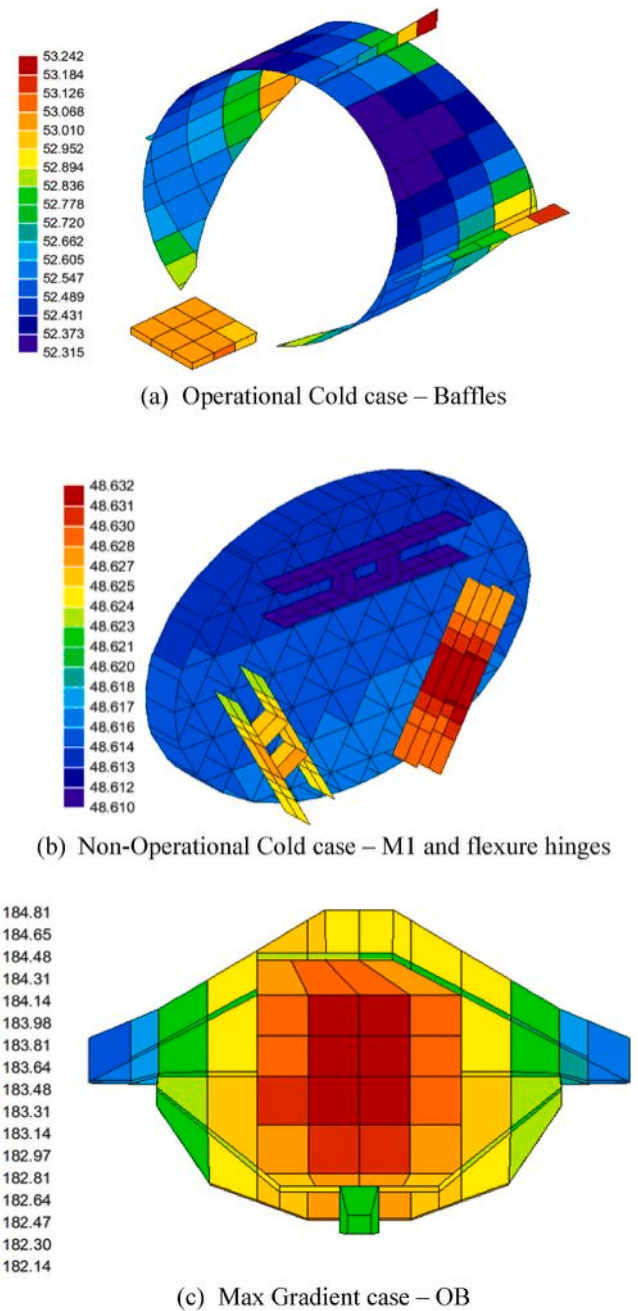


Fig. 20. Temperature maps (in Kelvin) obtained from ESATAN-TMS model on different assemblies.

temperature maps focusing on some relevant parts are shown in Fig. 20.

Once the temperatures for each load case have been calculated by the thermal analyses in ESATAN-TMS, the next step in the procedure depicted in Fig. 9 is the transfer of these temperatures to the FEM (temperature mapping). Two different methods have been exposed in section 2 and have been applied to compare their differences for the ARIEL PLM. For the PAT mapping method, the FEM thermal conductive analyses have been performed with a maximum of 500 independent nodes for the averaging MPCs. Additionally, the option for converting the structural rigid elements is the CELAS method (see Fig. 7 (right)).

The comparison of the resulting temperature maps transferred to the FEM between both temperature mapping methods for the three evaluated load cases is shown in Fig. 21 for the Telescope Assembly and in Fig. 22 for some relevant parts. It can be noticed on the baffle B1 the rectangular zones of constant temperature (patches) for the PWT

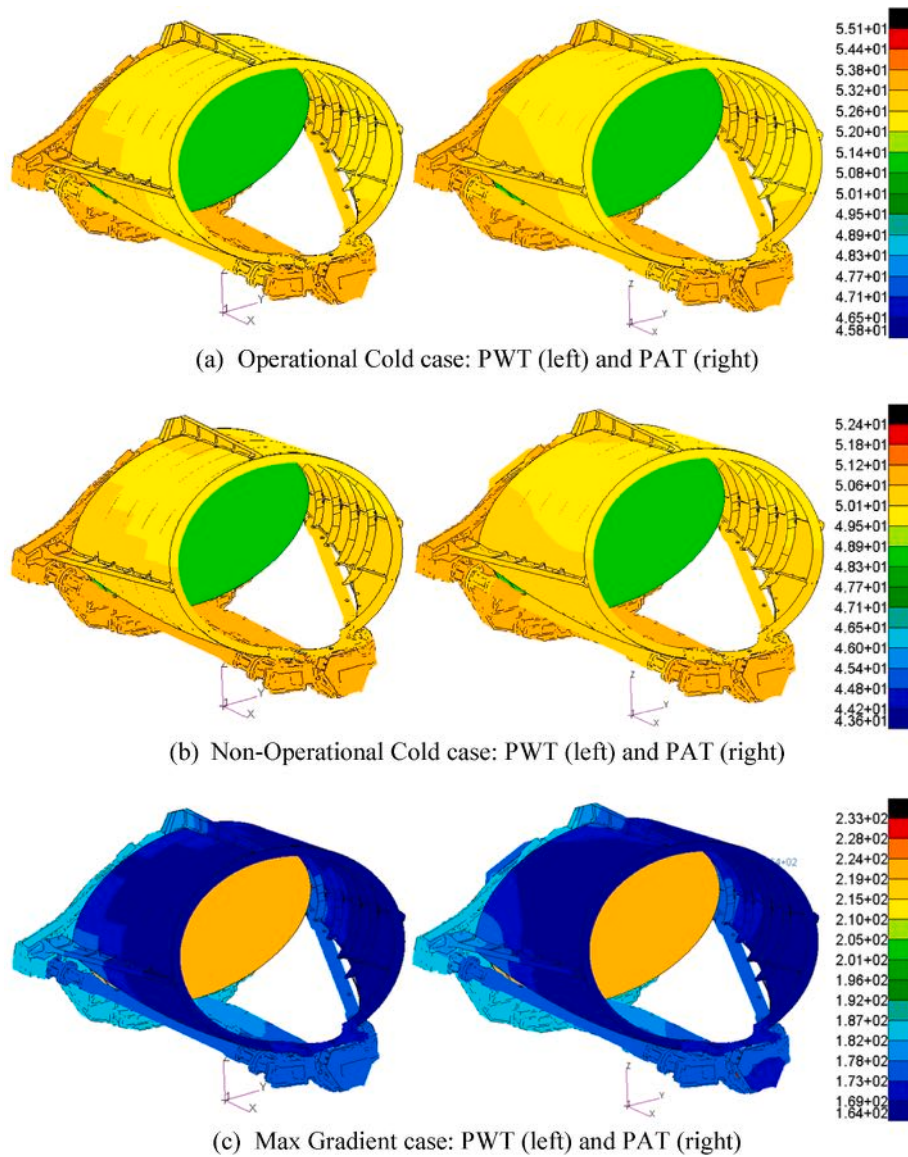


Fig. 21. Temperature maps (in Kelvin) transferred to the ARIEL Telescope Assembly FEM.

method, with temperature jumps between two adjacent zones. However, the PAT method provides smoother transitions of temperatures between the different zones. In general, the steady-state load cases (Operational and Non-Operational Cold cases) show a very uniform temperature map on the telescope assembly, where the difference between both temperature mapping methods is very small. However, the difference in the resulting temperature maps between both methods is greater for the transient load case (Max Gradient) due to its greater gradient. This can be appreciated in some local regions, such as for the M2 mirror and its mechanism (see Fig. 23).

Fig. 23 shows the comparison of the transferred temperature maps between PWT and PAT methods for the assembly of the mirror M2 and its mechanism for the Max Gradient case. For the partitioning of the models in the mapping process, M2 mirror and the M2M were in different groups. The high temperature jump between both parts (about 53 K) is noticeable in this transient state during the cool-down. The resulting temperature map obtained with the PWT method shows a jump just at the mechanical interface between the mirror and the mechanism, which in turn will produce very high shear forces on the screws for this interface, as will be shown below. However, for the PAT method, the thermal conductive analysis reveals that this temperature change

appears along the bipods that support the mechanical interfaces with the M2 mirror, reducing considerably the temperature jump between the M2 mirror and its interfaces, which in turn will reduce in the same order of magnitude the generated shear forces. This example shows one of the main differences between both mapping methods, which is the representation of the temperature transition along the interface between two parts with a significant gradient of temperature.

The next step in the STOP analysis procedure is the structural thermoelastic analysis (see Fig. 9), which calculates the generated deformation, stresses and forces caused by the temperature change between a uniform room temperature (293 K for these analyses) and a final temperature, previously calculated by the thermal model for the different load cases and transferred to the FEM by the two different options for the temperature mapping.

The objective of the operational load cases, such as the Operational Cold case, is the calculation of the deformations, translations, and rotations of the optical elements to evaluate the impact on the optical performance. One of the results directly calculated from the structural analysis that can give a quick assessment on the optical performance is the rigid body motion (RBM) of the main optical elements. The obtained RBM results for the Operational Cold case for the main mirrors M1 and

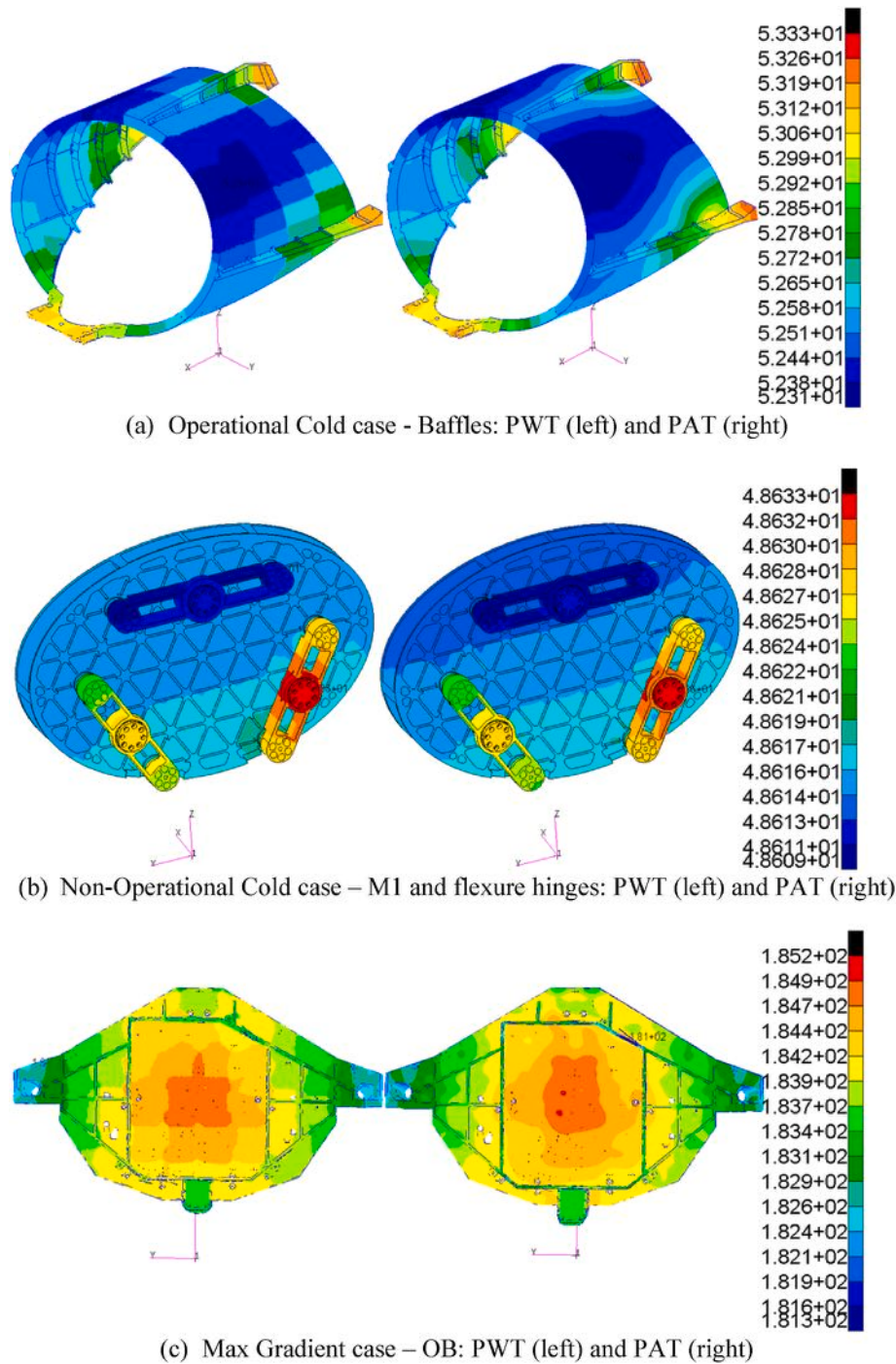


Fig. 22. Temperature maps (in Kelvin) transferred to the FEM on different assemblies.

M2 are shown in Table 5, comparing the results calculated with the input temperature maps from both mapping methods (PWT and PAT). Due to the great uniformity in the temperature map for the telescope assembly, the differences in the motions of the mirrors between both mapping methods are small. In this example, most of the relative differences are below 1 %, with the exception of the rotation of the M2 around the longitudinal X axis, with a difference of 2 % between both mapping methods. However, the very small absolute value of this rotation implies that the impact of the selected mapping method is practically negligible for this load case.

The main objective of the non-operational load cases is the calculation of the maximum stresses and forces on the different parts, considering these cases as severe scenarios from the structural point of view,

where the Non-Operational Cold case represents the steady-state situation with the coldest temperature, and the Max Gradient case represents a transient situation with a great gradient of temperature across the telescope assembly. The resulting maximum stresses calculated from the respective structural analyses are depicted in Table 6 for each main component. The stress distributions of the Max Gradient case are shown in Fig. 24, comparing the differences between the results from the two mapping methods.

In general, the comparison of the maximum stresses for the steady-state Non-Operational Cold case shows a great similarity between the results calculated with the temperature maps from both mapping methods. The small gradient of temperature across the telescope assembly for this steady-state case (see Fig. 21 (b)) explains these small

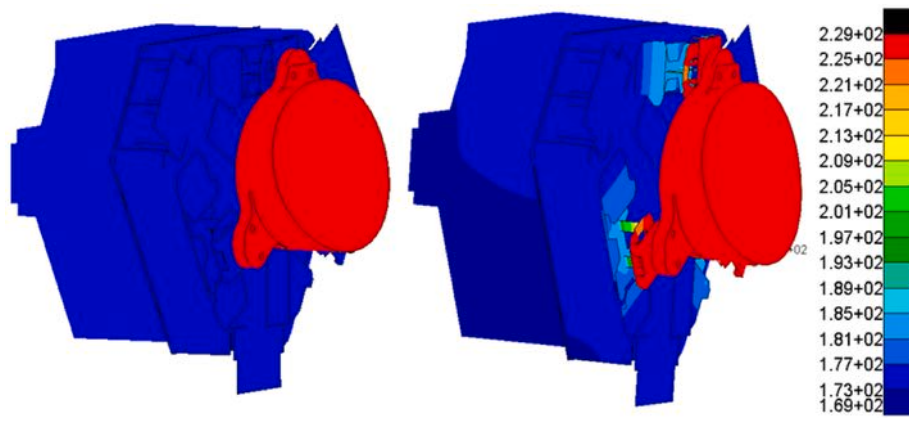


Fig. 23. Temperature maps (in Kelvin) on the FEM of the M2 mirror and its mechanism for Max Gradient case: PWT (left) and PAT (right).

Table 5

Rigid body translations (T_i) and rotations (R_i) of the mirrors M1 and M2 for the Operational Cold case.

Displacements	M1 mirror			M2 mirror		
	PWT	PAT	Difference	PWT	PAT	Difference
Tx [microns]	720.93	720.55	0.05 %	−3944.19	−3944.36	0.00 %
Ty [microns]	22.37	22.40	−0.10 %	29.81	29.81	0.00 %
Tz [microns]	−1381.56	−1378.48	0.22 %	−1136.85	−1134.20	0.23 %
Rx [μrad]	38.12	38.07	0.11 %	3.17	3.10	2.15 %
Ry [μrad]	1025.52	1023.17	0.23 %	594.45	594.64	−0.03 %
Rz [μrad]	−14.69	−14.72	−0.18 %	−10.71	−10.76	−0.44 %

Table 6

Maximum stresses calculated from the thermoelastic analysis of the ARIEL PLM FEM.

Part	Maximum stresses [MPa]			
	Non-Operational Cold case		Max Gradient case	
	PWT	PAT	PWT	PAT
Optical Bench	4.3	8.8	20.3	19.5
Metering structure	14.6	14.6	10.2	10.5
M1	19.3	19.3	26.2	27.2
M1 Supports	25.2	25.2	57.1	41.1
M2	0.8	0.8	53.6	0.3

differences in results between both mapping methods, in a similar way to the displacements results of the Operational Cold case. However, the appreciable gradient obtained for the transient Max Gradient case leads to greater differences in the structural results between both mapping methods in some regions, as can be appreciated for the supports of the M1 mirror and for the M2 mirror. It is particularly noticeable the big difference in stresses generated on the M2 mirror, where with the temperature map transferred by the PWT method, the big temperature jump between this component and the mechanism that supports it (see Fig. 23 (left)) provokes high stresses that reach around 54 MPa at the flanges (see Fig. 24 (e)). However, the thermal conductive analysis of the FEM included in the PAT method calculates a smoother temperature distribution, reducing considerably the temperature jump at the interface between the M2 mirror and its mechanism (see Fig. 23 (right)), generating negligible maximum stresses on the M2 (see Fig. 24 (f)), less than 1 MPa.

Regarding the generated forces on the joining elements (screws) represented by CBUSH elements, the results shown in Table 7 indicate the same consequences observed in the previous results, where the selection of the mapping method is not critical when analysing a load case with low gradient temperature, such as the Non-Operational Cold case, but has a significant impact on the results for load cases with high

gradients, such as the transient Max Gradient case. In this example, the shear forces of the joining elements between the M2 mirror and its mechanism are shown in Table 7 for the comparison of both mapping methods, where the numbering of the CBUSH elements can be seen in Fig. 14. Fig. 25 shows the distribution of the shear forces at this interface for the Max Gradient case with the temperature map transferred to the FEM by the PWT method. It is highlighted that the generated shear forces appear like pairs of opposite vectors for each flange, which is a consequence of the different thermoelastic contraction between these flanges and the supports of the mechanism connected to them.

This is the example found on the entire ARIEL PLM FEM with the maximum differences in the stresses and forces between the results of both mapping methods. But in general, because with the PWT method temperature jumps appear at the interfaces between the different parts, usually associated with different thermal elements of the ESATAN-TMS model, high stresses and forces are generated precisely in these interfaces. This fact produces the highest difference with respect to the results from the PAT method, where thanks to the FEM thermal conductive analysis, smoother transitions of temperature are calculated in the same interfaces, generating lower stresses and forces. Therefore, local results at the interfaces may be different depending on the selected mapping method, which is more evident for load cases with high gradient of temperature. Considering the better realism and accuracy of the PAT method compared to the PWT method, it is recommended the first one for this type of situations.

6. Conclusions

This work has the purpose of showing the successful application of two different temperature mapping methods on the real case of the Payload Module of the ARIEL spacecraft to demonstrate the feasibility of performing the proposed procedures for both methods. Some drawbacks such as the computational limitation for the thermal conductive analysis in NASTRAN with a huge FEM with large MPCs, and the difficulty of associating the nodes between the thermal and the structural models created with different modelling approaches have been solved in this

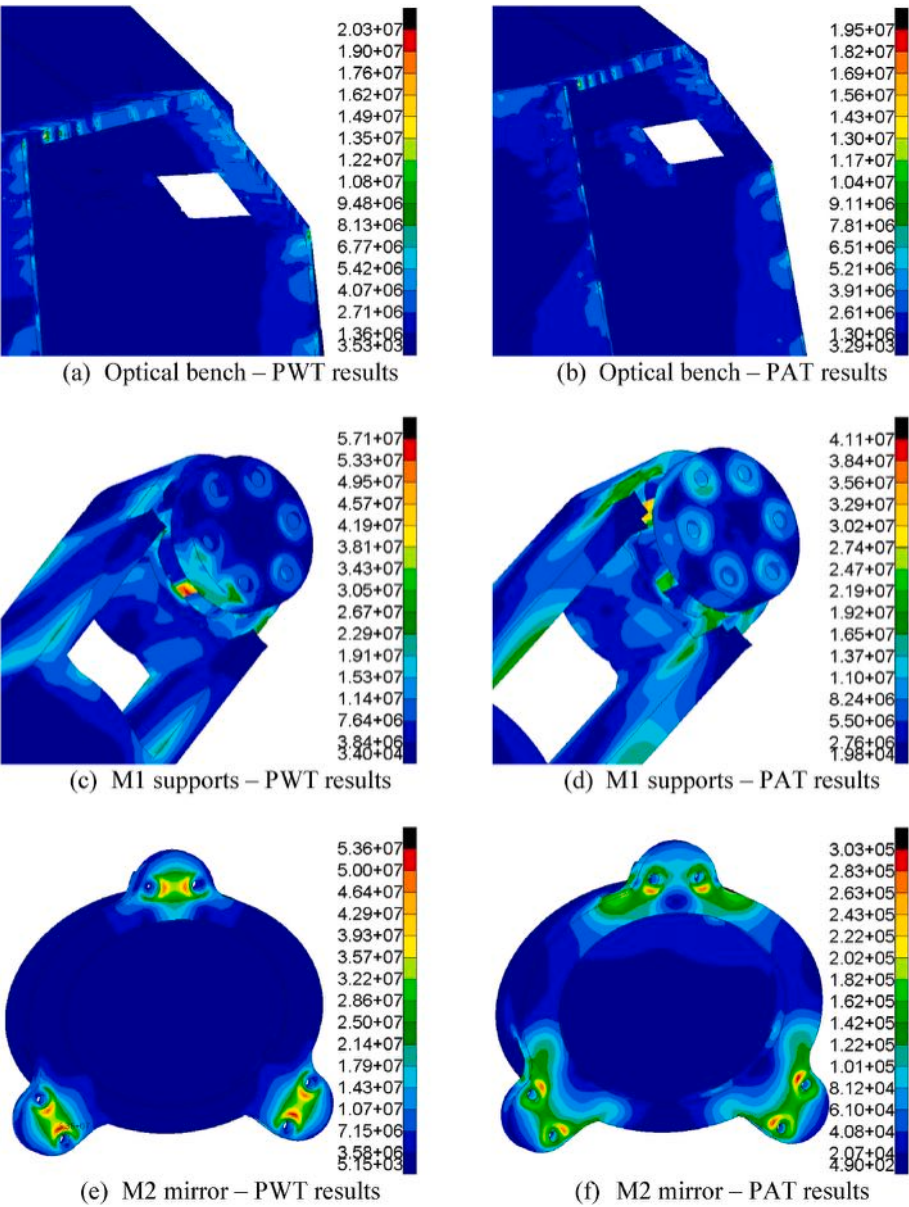


Fig. 24. Stress distribution (in Pascals) for the thermoelastic Max Gradient case.

Table 7
Shear forces generated at the M2 – M2M interface elements.

CBUSH ID	Non-Operational Cold case		Max Gradient case	
	Shear forces [N]		Shear forces [N]	
	PWT	PAT	PWT	PAT
5431	15.44	15.40	1863.92	9.40
5432	15.91	15.88	1864.23	9.47
5433	15.43	15.38	1861.62	9.96
5434	15.93	15.88	1861.92	10.01
5435	15.14	15.10	1881.34	8.60
5436	15.15	15.11	1881.33	8.58

work by the proposed methods.

Two existing temperature mapping methods (PWT and PAT) have been evaluated in this work, explaining in detail the proposed schemes for each one implemented in a dedicated software tool. Some particular processes have been implemented to improve the versatility of their use in complex models of any analysed system. One of the taken decisions

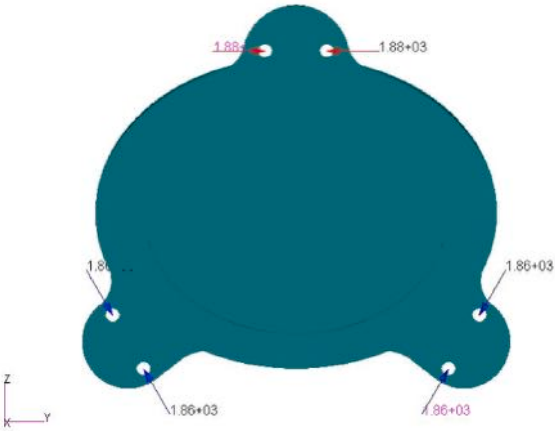


Fig. 25. Distribution of shear forces for the M2 mirror IF for the Max Gradient case with PWT mapping.

was the selection of the nearest-node search method to associate each structural node to its closest thermal node prior to the temperature transfer between both models, which avoids the problems and difficulties when working with models with different modelling approaches and mismatching between their geometries. In the example of the ARIEL PLM, the thermal model created in ESATAN-TMS is mainly composed of surface elements, while the structural FEM is created in detail with volumetric elements to represent the main parts. Therefore, the overlapping process between both models would become very difficult or even impossible when trying other more sophisticated methods that take into account the geometry shape of each element. Therefore, the selected nearest-node search technique is compatible with any difference in modelling approaches between both models. This allows structural and thermal engineers to work independently on their respective models to optimally achieve the best representation of real physics, by balancing adequately accuracy and computation resources. Another interesting aspect implemented for the proposed flowchart of the temperature mapping process is the possibility of dividing both models into different parts or groups, improving the temperature transfer between the corresponding parts and reducing the execution time during this process.

One of the key aspects studied in this work is the computational limit found when performing thermal conductive analyses in NASTRAN with models that contain several MPCs with a great quantity of independent nodes. The results obtained from this study reveal that the computational resources such as the required RAM memory and the usage of hard disk storage for the temporal files are practically proportional to the maximum number of independent nodes associated to each MPC in the analysed model. This is particularly severe for FEMs with millions of nodes, where all these nodes should be connected to MPCs necessary for the correct calculation of the temperature map with the PAT method version proposed in this work. The solution found in this work is the modification of the original set of large MPCs by set of smaller MPCs in a two-level tree configuration, which achieves the same accuracy in the results but with the significant reduction in the computational resources, making available the successful application of the PAT method in a complex FEM such as the ARIEL PLM.

The comparison of the results in terms of temperature maps, as well as the subsequent structural results such as displacements, stresses and forces between both temperature mapping methods for the ARIEL PLM concludes that the selection of the temperature mapping approach is not critical when analysing steady-state load cases with small gradients. Fortunately for the ARIEL mission, most of the load cases defined for the STOP analysis that involve the evaluation of the optical performance correspond to situations where the excellent thermal design ensures a very uniform temperature distribution throughout the entire telescope assembly, reducing possible thermoelastic distortions that can severely alter the correct operation of the astronomic observation. This level of uniformity of the temperature leads to a great similarity of the thermal and structural results between both mapping methods, preferring in these situations the PWT method due to its greater simplicity and speed. However, the selection of the mapping method becomes more relevant when analysis a load case with significant temperature gradients between some parts, causing remarkable differences between the calculated results. For these situations, the greater sophistication of the PAT method allows the calculation of more reliable results than those of the PWT method, being consequently recommended the PAT option if the time and computation constrains allow it.

Funding

This work has been funded by the Spanish ministry “Ministerio de Ciencia e Innovación” through project ARIEL_SCITECH with grant number PID2021-125627OB-C33.

CRedit authorship contribution statement

Andrés García-Pérez: Writing – review & editing, Writing – original draft, Software, Resources, Methodology, Investigation, Formal analysis, Data curation, Conceptualization. **Alejandro Fernández-Soler:** Writing – review & editing, Supervision, Software, Investigation, Formal analysis. **Gianluca Morgante:** Software, Project administration, Investigation, Formal analysis. **Javier Pérez-Álvarez:** Writing – review & editing, Visualization, Resources, Project administration, Conceptualization. **Gustavo Alonso:** Supervision, Project administration, Funding acquisition. **Laura García-Moreno:** Visualization, Resources. **Antonio Scippa:** Software, Resources, Investigation. **Daniele Gottini:** Software, Resources, Investigation. **Riccardo Lilli:** Software, Resources, Investigation.

Declaration of competing interest

The authors declare that they have no known competing financial interests or personal relationships that could have appeared to influence the work reported in this paper.

Acknowledgements

Authors would like to thank the overall ARIEL Consortium Team for this grateful collaborative work for the development of the Payload Module of the ARIEL spacecraft. We especially want to thank these partners (RAL, INAF, UniFi, Active Space, DTU, Sener, Admatis, CEA, CBK and JPL) for providing us with the different models of their corresponding subsystems to integrate them into the overall thermal and structural models of the ARIEL PLM, thereby achieving the completion of this work. Finally, we would like to recognise the excellent work performed by the ESA and RAL teams in managing this project.

References

- [1] M. Bourdeaud, J. Ponsy, S. Laborde, P. Corberand, Designing Satellites from the main thermo-elastic stability contributors quantification – process and tool applied on Juice, in: *Eur. Conf. Spacecr. Struct. Mater. Mech. Test.*, 2018.
- [2] M. Burkhalter-Lindner, W.V. Davies, P.J. Schöngel, B. Schmid, Structural design and verification of the Planck payload module structure, in: *Eur. Conf. Spacecr. Struct. Mater. Mech. Test.*, 2005. Noordwijk, The Netherlands.
- [3] V. Da Deppo, E. Pace, G. Morgante, M. Focardi, E. Pascale, G. Malaguti, M. Terraneo, F. Zocchi, G. Bianucci, G. Micela, The primary mirror of the ARIEL mission: study and development of a prototype, in: *Eur. Planet. Sci. Congr.*, 2018.
- [4] S. Fransen, D. Doyle, B. Catanzaro, Opto-mechanical modeling of the herschel space telescope at ESA/ESTEC, in: T. Andersen, A. Enmark (Eds.), *Integr. Model. Complex Optomech. Syst.*, 2011 833604, <https://doi.org/10.1117/12.915669>.
- [5] J.D. Johnston, J.M. Howard, G.E. Mosier, K.A. Parrish, M.A. McGinnis, A.M. Bluth, K. Kim, K.Q. Ha, Integrated modeling activities for the James Webb space telescope: structural-thermal-optical analysis, in: J.C. Mather (Ed.), *Opt. Infrared, Millim. Sp. Telesc.*, 2004, p. 600, <https://doi.org/10.1117/12.551704>.
- [6] M. Herceg, P.S. Jørgensen, J.L. Jørgensen, Characterization and compensation of thermo-elastic instability of SWARM optical bench on micro Advanced Stellar Compass attitude observations, *Acta Astronaut.* 137 (2017) 205–213, <https://doi.org/10.1016/j.actaastro.2017.04.018>.
- [7] S. Appel, J. Wijk, Simulation of Thermoelastic Behaviour of Spacecraft, Springer International Publishing, 2022, https://doi.org/10.1007/978-3-030-78999-2_8.
- [8] TEC-M ESA/ESTEC, European Guidelines for Thermo-Elastic Verification, 2023.
- [9] S.M. Merkowitz, S. Conkey, W.B. Haile, W.R. KellyIII, H. Peabody, P.J. Dumont, Structural, thermal, optical and gravitational modelling for LISA, *Class. Quantum Gravity* 21 (2004) S603–S610, <https://doi.org/10.1088/0264-9381/21/5/032>.
- [10] M. Levine, J. Fanson, Advanced thermo-structural technologies for the NASA terrestrial planet finder mission, *Struct. Control Heal. Monit.* 13 (2006) 190–209, <https://doi.org/10.1002/stc.136>.
- [11] S.J. Scola, J.F. Osmundsen, L.S. Murchison, W.T. Davis, J.M. Fody, C.M. Boyer, A. L. Cook, C.A. Hostetler, S.T. Seaman, I.J. Miller, W.C. Welch, A.R. Kosmer, Structural-Thermal-Optical-Performance (STOP) model development and analysis of a field-widened Michelson interferometer, in: G.G. Gregory, A.J. Davis (Eds.), *Nov. Opt. Syst. Des. Optim.*, vol. XVII, 2014, p. 91930I, <https://doi.org/10.1117/12.2061041>.
- [12] R. Gracey, A. Bartoszyk, E. Cofie, B. Comber, G. Hartig, J. Howard, D. Sabatke, G. Wenzel, R. Ohl, Structural, thermal, and optical performance (STOP) modeling and results for the James Webb Space Telescope integrated science instrument module, in: G.Z. Angeli, P. Dierickx (Eds.), *Model. Syst. Eng. Proj. Manag. Astron.*, VI, 2016, p. 99111A, <https://doi.org/10.1117/12.2233641>.

- [13] F. Navarro-Medina, A.E. Oudijk, M.B. Henriksen, U. Garcia-Luis, A. Gomez-San Juan, T.A. Johansen, Structural thermal optical performance (STOP) analysis and experimental verification of an hyperspectral imager for the HYPSON CubeSat, *Opt. Lasers Eng.* 173 (2024) 107919, <https://doi.org/10.1016/j.optlaseng.2023.107919>.
- [14] M. Cho, A. Corredor, K. Vogiatzis, G. Angeli, Thermal modeling of the TMT telescope, in: T. Andersen, A. Enmark (Eds.), *Integr. Model. Complex Optomech. Syst.*, 2011, p. 83360V, <https://doi.org/10.1117/12.915634>.
- [15] ECSS, ECSS-E-ST-31-04C, *Space engineering, Exchange of thermal analysis data, ESA-ESTEC Requirements and Standards Division*, Noordwijk, The Netherlands (2018).
- [16] ECSS, ECSS-E-ST-32-03C, *Space engineering, Structural finite element models. ESA-ESTEC Requirements and Standards Division*, 2008. Noordwijk, The Netherlands.
- [17] M. Koot, S. Appel, S. Simonian, Temperature mapping for structural thermo-elastic analyses ; method benchmarking, in: *Proc. Eur. Conf. Spacecr. Struct. Mater. Mech. Test.*, 2018. Noordwijk, The Netherlands.
- [18] M. Koot, S. Appel, S. Simonian, Thermal conductor generation for thermal and thermo-elastic analysis using a finite element model and sinas, in: *Proc. Eur. Conf. Spacecr. Struct. Mater. Mech. Test.*, Noordwijk, The Netherlands, n.d.
- [19] L.A. Tse, Z. Chang, R.P. Somawardhana, E. Slimko, Structural, thermal, and optical performance (STOP) modeling and analysis for the surface water and ocean topography mission, in: *48th Int. Conf. Environ. Syst.*, 2018. Albuquerque, NM.
- [20] E. Heesel, T. Weigel, P. Lochmatter, E. Rugi Grond, Coupled thermo-elastic and optical performance analyses of a reflective baffle for the BepiColombo laser altimeter (BELA) receiver, in: J. Costeraste, E. Armandillo, N. Karafolas (Eds.), *Int. Conf. Sp. Opt. — ICSO 2008*, SPIE, 2017, p. 68, <https://doi.org/10.1117/12.2308252>.
- [21] C. Blaurock, M. McGinnis, K. Kim, G.E. Mosier, Structural-thermal-optical performance (STOP) sensitivity analysis for the James Webb space telescope, in: M. A. Kahan (Ed.), *Opt. Model. Perform. Predict*, II, 2005, p. 58670V, <https://doi.org/10.1117/12.618697>.
- [22] S. Appel, SINAS IV User ' S Manual, Fokker Space B.V, Leiden (The Netherlands), 1998.
- [23] S. Appel, *Using Pysinas*, 2024.
- [24] V. Da Deppo, M. Focardi, K. Middleton, G. Morgante, E. Pascale, S. Grella, E. Pace, R. Claudi, J. Amiaux, J. Colomé Ferrer, T. Hunt, M. Rataj, C. Sierra-Roig, I. Fical Veltroni, P. Eccleston, G. Micela, G. Tinetti, An afocal telescope configuration for the ESA ARIEL mission, *CEAS Sp. J.* 9 (2017) 379–398, <https://doi.org/10.1007/s12567-017-0175-3>.
- [25] E. Pascale, N. Bezawada, J. Barstow, J.-P. Beaulieu, N. Bowles, V. Coudé du Foresto, A. Coustenis, L. Decin, P. Drossart, P. Eccleston, T. Encrenaz, F. Forget, M. Griffin, M. Güdel, P. Hartogh, A. Hesse, P.-O. Lagage, J. Leconte, P. Malaguti, G. Micela, K. Middleton, M. Min, A. Moneti, J.C. Morales, M. Ollivier, E. Pace, G. Pilbratt, L. Puig, M. Rataj, T. Ray, I. Ribas, M. Rocchetto, S. Sarkar, F. Selsis, W. Taylor, J. Tennyson, G. Tinetti, D. Turrini, B. Vandenbussche, O. Venot, I. P. Waldmann, P. Wolkenberg, G.S. Wright, M.-R. Zapatero Osorio, T. Zingales, A. Papageorgiou, L. Mugnai, The ARIEL space mission, in: H.A. MacEwen, M. Lystrup, G.G. Fazio, N. Batalha, E.C. Tong, N. Siegler (Eds.), *Sp. Telesc. Instrum. 2018 Opt. Infrared, Millim. Wave, SPIE*, 2018, p. 16, <https://doi.org/10.1117/12.2311838>.
- [26] E. Pace, A. Tozzi, M. Adler Abreu, G. Alonso, B. Barroqueiro, G. Bianucci, A. Bocchieri, D. Brienza, A. Brucalassi, M. Burresi, R. Canestrari, L. Carbonaro, J. Castanheira, P. Chioetto, J. Colomé Ferrer, C. Compostizo, F. Cortecchia, F. D'Anca, C. Del Vecchio, E. Diolaiti, P. Eccleston, S. Fahmy, A. Fernandez Soler, D. Ferruzzi, M. Focardi, S. Freitas, C. Galy, A. Garcia Perez, D. Gottini, S. Grella, G. Groni, E. Guerriero, J.-P. Halain, M.-L. Hellin, L. Ianni, M. Iuzzolino, D. Jollet, M. Lombini, R. Machado, G. Malaguti, A. Mazzoli, G. Micela, F. Miceli, G. Mondello, G. Morgante, L. Mugnai, L. Naponiello, V. Noce, E. Pascale, J. Perez Alvarez, R. Piazzolla, C. Pompei, G. Preti, S. Roose, M. Salatti, J.-C. Salvignol, A. Scippa, C. Serre, C. Simoncelli, F. Teixeira, L. Terenzi, G. Tinetti, L. Tommasi, E. Tommasi Di Vigano, B. Vandenbussche, D. Vernani, P. Zuppella, The telescope assembly of the Ariel space mission, in: L.E. Coyle, M.D. Perrin, S. Matsuura (Eds.), *Sp. Telesc. Instrum. 2022 Opt. Infrared, Millim. Wave, SPIE*, 2022, p. 36, <https://doi.org/10.1117/12.2629432>.
- [27] P. Chioetto, A. Tozzi, A. Brucalassi, D. Ferruzzi, A. Caldwell, M. Caldwell, F. Cortecchia, E. Diolaiti, P. Eccleston, E. Guerriero, M. Lombini, G. Malaguti, G. Micela, E. Pace, E. Pascale, R. Piazzolla, G. Preti, M. Salatti, G. Tinetti, E. Tommasi, P. Zuppella, Preliminary analysis of ground-to-flight mechanical tolerances of the Ariel mission telescope, in: L.E. Coyle, M.D. Perrin, S. Matsuura (Eds.), *Sp. Telesc. Instrum. 2022 Opt. Infrared, Millim. Wave, SPIE*, 2022, p. 220, <https://doi.org/10.1117/12.2628900>.
- [28] G. Alonso, A. Garcia-Perez, J. Perez-Alvarez, A. Fernandez-Soler, The mechanical , structural and thermal design of the Telescope Assembly of ARIEL, in: *Proc. Eur. Sci. Congr.* 2022, 2022, pp. 16–18, <https://doi.org/10.5194/eps2022-861>. Granada, Spain.
- [29] A. Tozzi, A. Brucalassi, R. Canestrari, P. Chioetto, C. Del Vecchio, L. Carbonaro, F. Cortecchia, E. Diolaiti, P. Eccleston, G. Falcini, D. Ferruzzi, D. Gottini, E. Guerriero, M. Iuzzolino, R. Lilli, M. Lombini, G. Malaguti, G. Micela, F. Miceli, G. Morgante, E. Pace, E. Pascale, R. Piazzolla, G. Preti, M. Salatti, A. Scippa, G. Tinetti, E. Tommasi, D. Vernani, P. Zuppella, Toward ARIEL's primary mirror, in: L.E. Coyle, M.D. Perrin, S. Matsuura (Eds.), *Sp. Telesc. Instrum. 2022 Opt. Infrared, Millim. Wave, SPIE*, 2022, p. 193, <https://doi.org/10.1117/12.2628906>.
- [30] P. Chioetto, P. Zuppella, V. Da Deppo, E. Pace, G. Morgante, L. Terenzi, D. Brienza, N. Missaglia, G. Bianucci, S. Spinelli, E. Guerriero, M. Rossi, C. Grèzes-Besset, C. Bondet, G. Chauveau, C. Porta, G. Malaguti, G. Micela, Qualification of the thermal stabilization, polishing and coating procedures for the aluminum telescope mirrors of the ARIEL mission, *Exp. Astron.* 53 (2022) 885–904, <https://doi.org/10.1007/s10686-022-09852-x>.
- [31] E. Guerriero, P. Chioetto, A. Tozzi, P. Zuppella, R. Canestrari, A. Brucalassi, M. Iuzzolino, D. Ferruzzi, A. Scippa, C. Del Vecchio, G. Falcini, L. Carbonaro, G. Morgante, F. Cortecchia, E. Diolaiti, P. Eccleston, M. Lombini, G. Malaguti, G. Micela, E. Pace, E. Pascale, R. Piazzolla, G. Preti, M. Salatti, G. Tinetti, E. Tommasi, Heat treatment procedure of the aluminium 6061-T651 for the Ariel telescope mirrors, in: L.E. Coyle, M.D. Perrin, S. Matsuura (Eds.), *Sp. Telesc. Instrum. 2022 Opt. Infrared, Millim. Wave, SPIE*, 2022, p. 39, <https://doi.org/10.1117/12.2628178>.
- [32] D. Gottini, E. Pace, A. Tozzi, G. Bianucci, A. Bocchieri, D. Brienza, A. Brucalassi, R. Canestrari, L. Carbonaro, P. Chioetto, F. Cortecchia, C. Del Vecchio, E. Diolaiti, P. Eccleston, S. Fahmy, D. Ferruzzi, C. Galy, G. Groni, E. Guerriero, J.-P. Halain, M.-L. Hellin, M. Iuzzolino, D. Jollet, M. Lombini, G. Malaguti, G. Micela, N. Missaglia, G. Morgante, L. Mugnai, L. Naponiello, E. Pascale, R. Piazzolla, G. Preti, S. Roose, M. Salatti, J.-C. Salvignol, A. Scippa, L. Terenzi, G. Tinetti, E. Tommasi Di Vigano, P. Zuppella, FEA testing the pre-flight Ariel primary mirror, in: L.E. Coyle, M.D. Perrin, S. Matsuura (Eds.), *Sp. Telesc. Instrum. 2022 Opt. Infrared, Millim. Wave, SPIE*, 2022, p. 195, <https://doi.org/10.1117/12.2629815>.
- [33] G. Morgante, L. Terenzi, D. D'Ascanio, P. Eccleston, M. Crook, T. Hunt, V. Da Deppo, M. Focardi, G. Micela, G. Malaguti, E. Pace, G. Tinetti, Thermal architecture of the ESA ARIEL payload, in: H.A. MacEwen, M. Lystrup, G.G. Fazio, N. Batalha, E.C. Tong, N. Siegler (Eds.), *Sp. Telesc. Instrum. 2018 Opt. Infrared, Millim. Wave, SPIE*, 2018, p. 154, <https://doi.org/10.1117/12.2313153>.
- [34] G. Morgante, L. Terenzi, L. Desjonqueres, P. Eccleston, G. Bishop, A. Caldwell, M. Crook, R. Drummond, M. Hills, T. Hunt, D. Rust, L. Puig, T. Tirolien, M. Focardi, P. Zuppella, W. Holmes, J. Amiaux, M. Berthé, C. Cara, C. Delisles, A. Direk, L. Dumaye, J. Fontignie, A. Goetschy, B. Horeau, N. Hurtado, D.-D. Huyn, G. Kaszubiak, P.-O. Lagage, I. Le Mer, M. Lortholary, V. Moreau, S. Mouzali, P. Mulet, F. Nico, T. Pichon, L. Provost, D. Renaud, V. Schwartz, M. Talvard, T. Tourrette, F. Visticot, A. Arhancet, D. Bachet, N. Berton, M. Lacroix, H. Le Provost, O. Tellier, A. Philippon, C. De Jabrun, J.-P. Dubois, F. Langlet, D. Le Claire, B. Lecomte, M. Ollivier, S. Tosti, V. Lapeyrière, M. Bonafous, J. Parisot, J.-M. Réess, D. Zegadnanin, J.-P. Beaulieu, V. Batista, P. Drossart, S. Fahmy, D. Jollet, L. Puig, T. Tirolien, P. Danto, G. Hervet, O. Maisonnave, P. Eccleston, G. Bishop, R. Drumond, A. Caldwell, M. Caldwell, L. Desjonqueres, M. Whalley, E. Pascale, G. Morgante, M. Focardi, E. Pace, A.-M. Di Giorgio, AIRS: ARIEL IR spectrometer status, in: L.E. Coyle, M.D. Perrin, S. Matsuura (Eds.), *Sp. Telesc. Instrum. 2022 Opt. Infrared, Millim. Wave, SPIE*, 2022, p. 37, <https://doi.org/10.1117/12.2628920>.
- [37] A. García-Pérez, G. Alonso, A. Gómez-San-Juan, J. Pérez-Álvarez, Thermoelastic evaluation of the payload module of the ARIEL mission, *Exp. Astron.* 53 (2022) 831–846, <https://doi.org/10.1007/s10686-021-09704-0>.
- [38] J.N. Loughhead, G.A. Miles, J.R. Turner, The ESATAN thermal network analysis computer program. <https://doi.org/10.4271/840967>, 1984.
- [39] V.L. Genberg, G.J. Michels, Integrated optomechanical STOP analysis linking nastran to Zemax, CODEV, and OSLO with SigFit (2017) 1–27.
- [40] A. Fernández-Soler, G. Morgante, A. García-Pérez, J. Pérez-Álvarez, Preliminary ariel decontamination thermal analysis, in: *Eur. Sp. Therm. Eng. Work.*, 2023.
- [41] S. Tuttle, L. Vaillon, U. Johann, O. Wallner, K. Ergenzinger, EUCLID mission design, in: N. Kadowaki (Ed.), *Int. Conf. Sp. Opt. — ICSO 2010*, 2017, p. 90, <https://doi.org/10.1117/12.2309226>. SPIE.