Application of the Substructuring Technique to the Stress Analysis of a Railcar Underframe Bolster

J. Fernández (*), M.A. Jiménez (*), J.M. Pradera (**) and S. Lafuente (**)

(*) Instituto Tecnológico de Aragón; Mechanics & New Materials Area

(**) CAF, S.A.; Structures Area; Technical Office of Zaragoza

Abstract: The bolsters are the principal transverse structural members of a railcar underframe. One is located near each end directly over the railcar bogie. Railcar bogies are very important in safe railway operations since, among other functions, are the components in charge of ensuring a good ride comfort by absorbing the vibrations generated by track irregularities and minimizing impact of centrifugal forces when train runs on curves at high speed. Connection from the bolster to the railcar bogie is via a special bolted bracket made of alloy steel casting that exerts the function of load transfer between both components. Due to the type of train (suburban train) to which the bolster belongs, under service conditions there are frequent acceleration-braking cycles that produce fatigue loads which effects on the bolster are of primary interest (in such cycles appear the maximum load amplitudes). This effect is less important in components belonging to long distance trains because of the lower number of stops along the journey.

The main goal of the work described in this paper was to obtain the detailed stress distributions in some regions of the bolster (welded joints) in order to assess the fatigue life of the component. To achieve such objective it was essential to model as exactly as possible the interaction between the different parts involved in the bolted joint to the bracket. Thus, contact elements were used to simulate the interaction and pre-tension was applied to the bolts.

1. Introduction

The present paper describes the structural analysis carried out on a bolster that belongs to the underframe of a suburban train, designed and manufactured by the company CAF, S.A.

Bolsters are the principal transverse structural elements of the railcar underframes. They are usually located in the ends of such underframes, just upon the railcar bogies (see Figure 1).

The main components of the analyzed bolster are two aluminium extruded beams joined together by means of welded aluminium plates of different thicknesses. This bolster is linked to the bogie by means of a special bracket that exerts the function of load transfer between both parts. The joint between the front plate of the bolster and the bracket is via six M36 bolts.

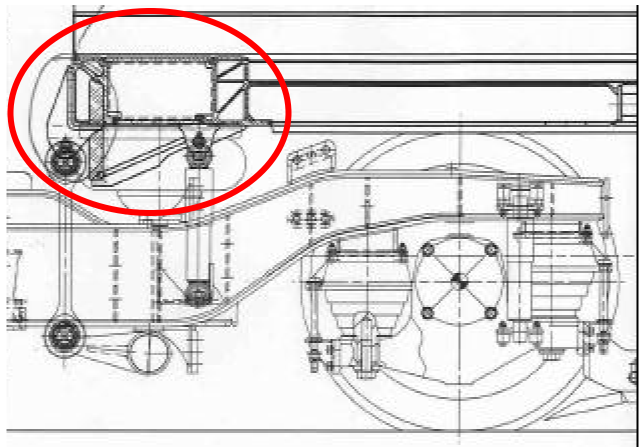

Figure 1. Situation of the bolster into the railcar underframe.

The aluminium alloys employed in the manufacturing of the bolster belongs to the 6000 series and their properties are included in the UNE EN 485-2 and UNE EN 755-2 standards. In the finite element models it will be used a linear, elastic, homogeneous and isotropic material model for the aluminium with the mechanical properties included in the table 1.

Table 1. Mechanical properties of the aluminium.

YOUNG MODULUS (MPa)	POISSON COEFFICIENT
70000	0.3

2. Calculation Model

2.1 Planning the model

At the beginning of the work, it was necessary to take some decisions involved in the characteristics of the model to be developed that were directly related with the analysis specifications. These specifications were the following:

- The model had to simulate as accurate as possible the global stiffness of the system bodyshell-underframe and it was important to include the effect of such elements in the structural behavior of the bolster. A finite element mesh including the bodyshell, the underframe and a simplified bolster was available from previous analysis (see Figure 2).
- With the aim of simulating in a proper way the loads transferred from the bogie to the bolster, the model must also include the special bracket that acts as load transmissor between both parts. Again, a detailed finite element mesh of the bracket was available for its use in the new analysis (Figure 3).
- One of the most important features of the structural system was the bolted joint between the bolster and the bracket, so it was essential to simulate it as real as possible, taking into account both the pre-tension of the bolts and the contact interaction between the different parts involved in that joint.
- As the bolster lies on the secondary pneumatic suspension system of the train, it was also necessary to include the effect of such suspension in the analysis. It was decided to assume a linear response of the pneumatic suspension components and to use an approximated value for the stiffness of such suspension.

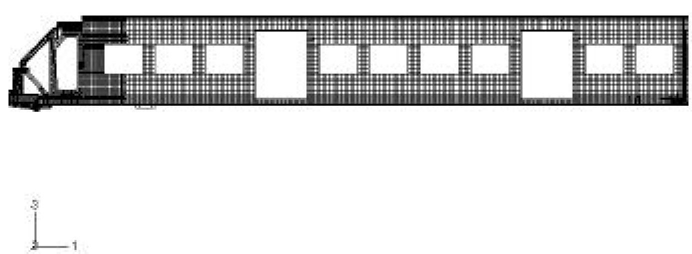

Figure 2. Finite element mesh of the bodyshell-underframe structure.

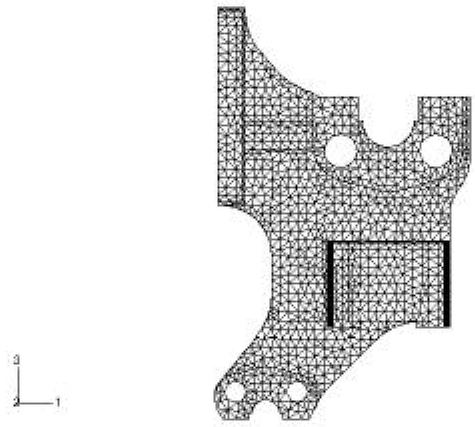

Figure 3. Finite element mesh of the special bracket.

The first of the prior specifications could be satisfied by using two advanced analysis techniques, namely, *submodelling* and *substructuring*.

On the one hand if the choice was submodelling, the analysis procedure was the following: to calculate the complete model including bodyshell + underframe + simplified bolster (using the available mesh), to obtain the nodal displacements in all nodes belonging to the boundary between the bodyshell/underframe and the bolster, to build up a detailed model of the bolster and to calculate it applying the nodal displacements coming from the previous analysis of the global model as boundary conditions. There was one drawback with this choice: the mesh of the bodyshell/underframe was not very fine while that one of the bolster had to be as fine as possible, under reasonable limits. The nodal displacements obtained in the global model, when applied as boundary conditions to the detailed bolster model, had to be interpolated in all regions of such model with existing nodes that were not present in the global model. We had some doubts about the level of accuracy that this displacements interpolation process could offer as a substitute of the real restrictions exerted by the bodyshell/underframe structure on the bolster.

On the other hand, the use of the substructuring technique implied the definition of the bodyshell/underframe set and the special bracket as *superelements*, and the later analysis of the detailed bolster model together with them. Again, the drawback of the different mesh density of the superelements and the detailed bolster added difficulties to the simulation of the interaction between the different components of the model.

Once the advantages and drawbacks of both techniques were clear, it was decided to use substructuring as the analysis technique. In this choice played a principal role the conviction of obtaining a better approximation to the global stiffness of the system by using substructuring than using submodelling. Also, substructuring could help to save disk space and computing time taking into account the non linear character of the analysis and the previewable large size of the model.

2.2 Substructuring

Substructuring is a procedure that condenses a group of finite elements into one element represented as a matrix. This single matrix element is called a superelement. You can use a superelement in an analysis just as you would use any other ABAQUS element type. The only difference is that you first create the superelement by performing a substructure generation analysis.

The usual reasons to use substructuring are related well to the savings of computing time well to the solution of very large problems. Examples of the these reasons are nonlinear analyses and analyses of structures that contain repeated geometrical patterns. In a nonlinear analysis you can substructure the linear portion of the model so that the element matrices for that portion might not be recalculated every equilibrium iteration. Also in a structure with repeated patterns you can generate one superelement to represent the pattern and simply make copies of it on different locations thereby saving a significant amount of computer time.

In the case presented in this paper both, the bodyshell-underframe structure (except for the bolster) and the special bracket, have been defined as superelements while the bolster is the main structure inside the global model. The summarized stages sequence of the analysis carried out is as follows:

- *Generation of the substructure associated to the bodyshell-underframe set.* This task is performed as an independent analysis. The model includes only the bodyshell and the underframe except for the bolster that will be modelled in detail later. At this stage it is very important to remember that all nodal degrees of freedom where boundary conditions or contact pairs may have to be applied during usage of the substructure in the global analysis must be *retained*.
- *Generation of the substructure associated to the special bracket.* The procedure carried out is identical to that of the bodyshell-uderframe set.
- *Global analysis.* In this analysis the model includes the detailed finite element mesh of the bolster and the two prior generated superelements. All loads, restrictions and contact pairs applied to this model are explained in detail later.

2.3 Model Description

Figure 4 shows the finite element models of the bodyshell-underframe set, the special bracket and the bolster with an indication of their role in the global analysis and the assembly scheme. As it can be seen in the figure, we have taken advantage of the symmetry of the system (both, geometric and load symmetry) for modeling only half of the different components, with the subsequent savings in disk space and computing time.

Figure 5 shows a detail of the bolster assembled in the bodyshell and already joined to the special bracket by means of the bolts. Finally, figure 6 shows the detailed finite element mesh of the bolster, isolated from the rest of components.

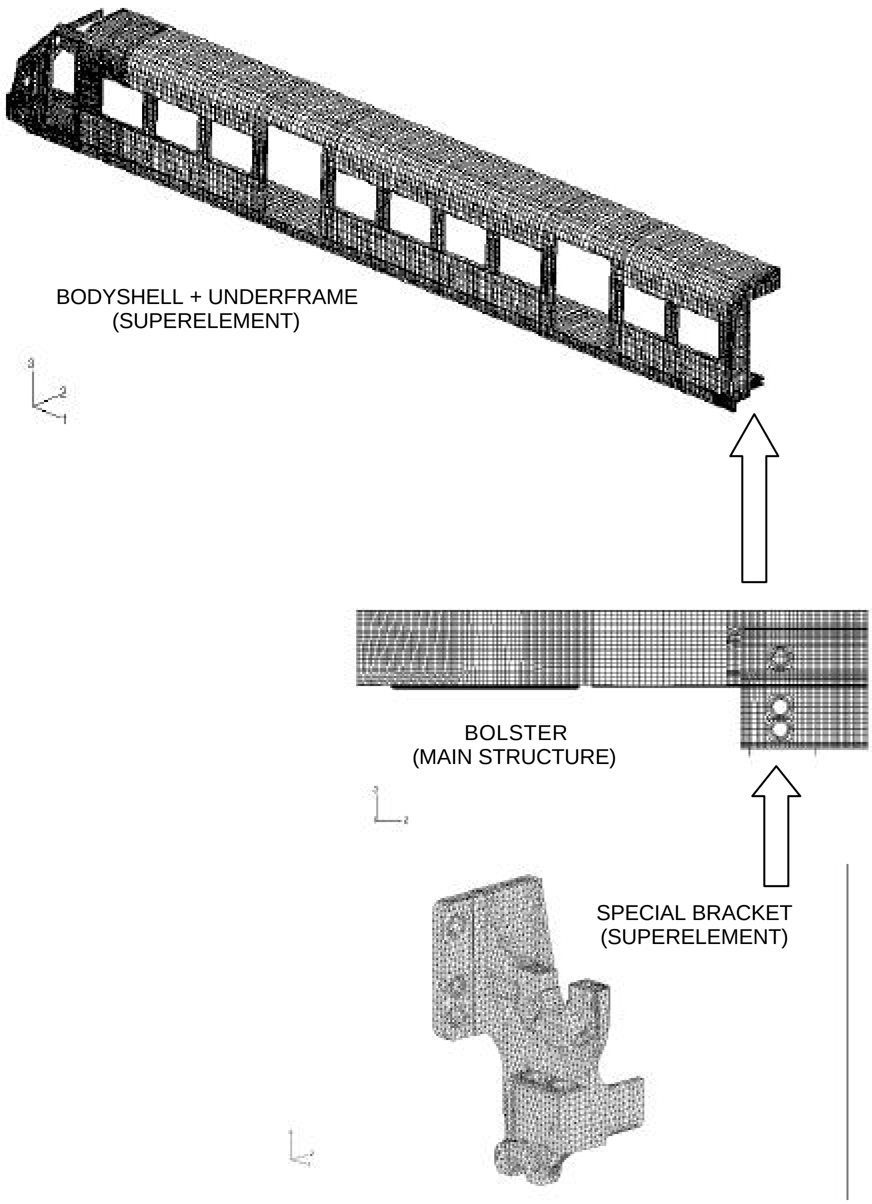

Figure 4. Assembly scheme of the analyzed model.

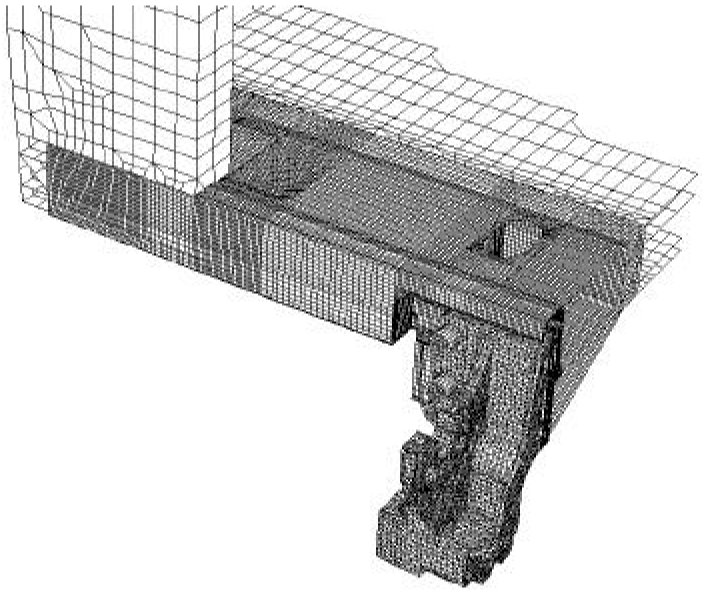

Figure 5. The bolster assembled together with the bracket and the bodyshell.

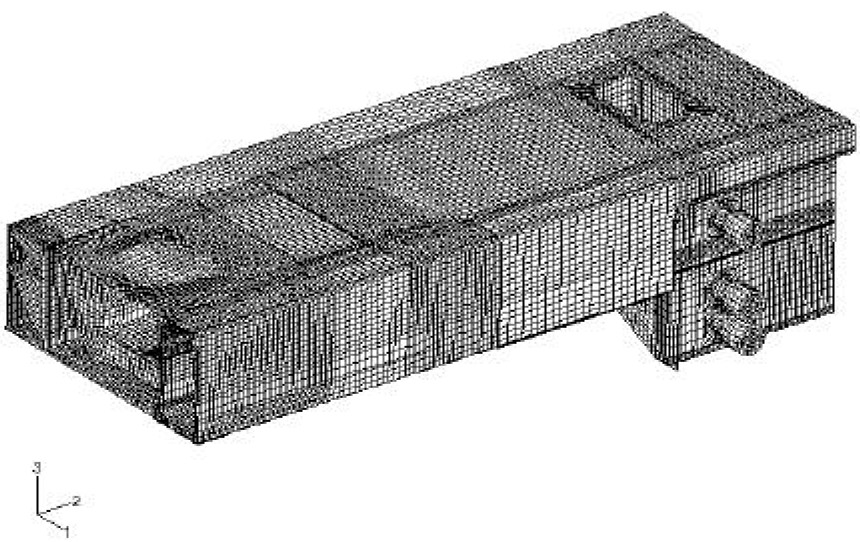

Figure 6. Detailed model of the bolster.

In the following the different finite element models included in the analysis are described in detail.

1. Bodyshell – underframe model.

   - S3R and S4R shell elements have been used to model all plates belonging to the bodyshell and the undeframe.

   - B31 beam elements to model all reinforcement bars existing in some zones of the bodyshell.

2. Special bracket model.

   - C3D4 solid elements have been used to model the body of the bracket. Although the use of linear tetrahedrons is not recommended for stress analysis in a general case, in this case the bracket acts like a superelement and so the usage of the C3D4 element is admissible.

3. Detailed bolster model.

   - C3D6 and C3D8R solid elements have been used to model the front main aluminium profile and the front support plate of the bolster. As can be seen in the Figure 6 a mapped mesh has been built for almost all regions of this components, so most of the elements are C3D8R and only a few C3D6 elements were used in some regions with geometric irregularities.

   - C3D8 solid elements are used to model the three M36 bolts that join the bolster to the special bracket.

   - S3R and S4R shell elements are used to model the rest of the aluminium plates and profiles of the bolster.

   - SPRING1 elements are used to simulate the effect of the pneumatic suspension of the train. Such suspension is simulated by means of a number of this elements located in the lower plate of the bolster. Figure 7 shows a simplified scheme of the suspension model used in the analysis.

   - M3D4R membrane elements are used to generate an auxiliar surface that allow to establish properly the contact pairs between one of the ends of the bolster and the underframe.

   - Two *superelements* corresponding to the bodyshell – underframe set and the special bracket. This elements are identified in the input file as belonging to the types Z10 and Z11 following the ABAQUS notation for substructures.

The table 1 shows the total number of nodes and elements used in the global model of the bolster.

Table 1. Number of nodes and elements included in the detailed bolster model.

NODE NUMBER	ELEMENT NUMBER
113046	92929

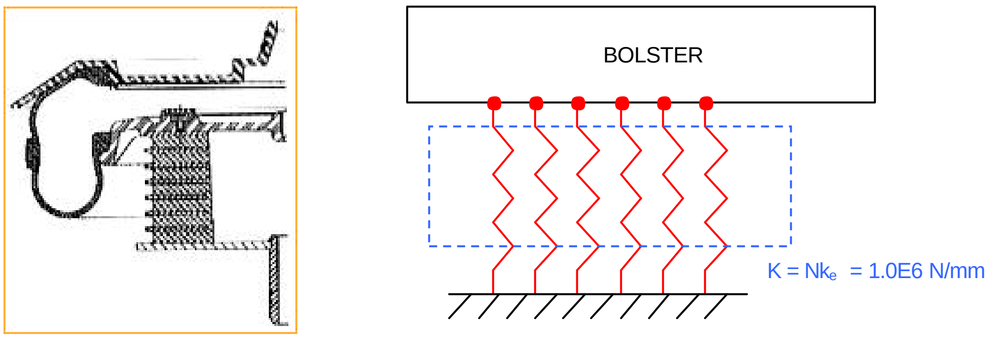

Figure 7. Real pneumatic spring and simulation model used in the analysis.

2.4 Boundary Conditions

Next, we are going to describe the boundary conditions applied to the global model. As already commented, when the boundary conditions are to be applied to a superelement all nodal degrees of freedom object of such boundary conditions must be previously retained in the substructure generation analysis.

2.4.1 Bodyshell – underframe

- Displacements in the longitudinal direction of the train (1 direction in the Figure 4) of all nodes belonging to the front plate of the bodyshell are set to zero.
- Displacement in vertical direction of the node located in the position of the forward pneumatic suspension of the train is set to zero.
- Symmetry conditions are applied to all nodes inside the symmetry plane of the model.

2.4.2 Special bracket.

- Symmetry conditions are applied to all nodes inside the symmetry plane of the model.

2.4.3 Bolster

- Displacements in the longitudinal and transversal directions of the train (1 &2 directions in the Figure 4) of all nodes belonging to the SPRING1 elements that simulate the back suspension of the train are set to zero. The objective of this prescribed condition is to avoid the free movement of such elements due to the lack of stiffness in the aforementioned directions.

- Rotations in all spatial directions are set to zero for all nodes common to shell and solid elements. Although ABAQUS incorporates tools to correct the uncertain behavior associated with the transition between shell and solid elements (shell-to-solid couplings) we have directly eliminated the rotation DOF of all nodes located in the transition zones. The validity of this technique was previously checked by means of several proof analyses.

- Rotations in all spatial directions are set to zero for all nodes of the S3R and S4R elements involved in the *tied* type contact pairs defined between some zones of the bodyshell and the bolster. Once again, this prescribed condition is intended to avoid the potential hinges that could appear in that contact regions.

- Finally, symmetry conditions are applied to all nodes inside the symmetry plane of the model.

2.5 Contact interactions

With the aim of simulating in the most possible realistic way the interaction between the different parts, contact pairs have been defined in the real contact zones between components. These contact pairs are of different type depending on the involved components. *Tied* contact has been used when the surfaces involved were thought as probably always remaining in contact along the analysis. When the previous condition could not always be satisfied, the contact pair was defined as *small sliding* with a friction coefficient of 0.1.

Next, the different contact pairs defined in the model are described in detail:

- Bolts – Special bracket (tied)
- Support plate of the bolster – Special Bracket (small sliding, $\mu = 0.1$)
- Bolts – Back zone of the front beam of the bolster (small sliding, $\mu = 0.1$)
- Underframe – Back beam of the bolster (tied)
- Underframe – End of the bolster (tied)
- Bodyshell – Top of the bolster (tied)
- Support plate of the bolster - Back zone of the front beam of the bolster (small sliding, $\mu = 0.1$)

2.6 Loads and load cases

Three load cases were considered in the analysis: pre-tension of the bolts, braking action and acceleration action. The pre-tension of the bolts was maintained during the application of the other load cases, thus achieving a good approximation to the real work conditions of the bolster.

2.6.1 Pre-tension of the bolts

An axial load was applied to each one of the bolts included in the model (see Figure 8) following the ABAQUS procedure for applying pre-tension loads on bolts.

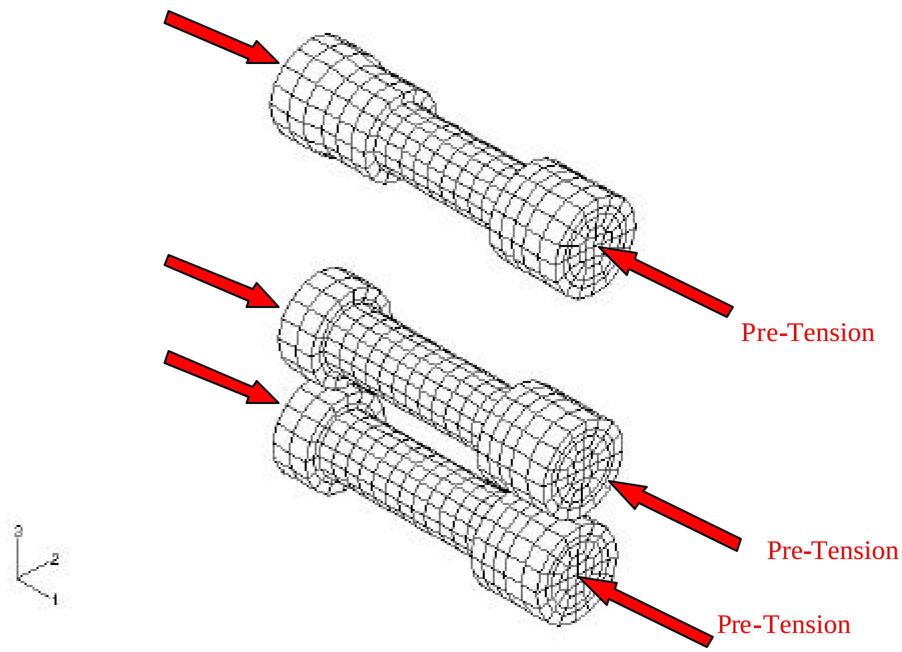

Figure 8. Pre-tension loads applied to the bolts.

2.6.2 Acceleration action

When the train starts his movement or increases its speed, the special bracket tries to separate itself from the bolster. This effect was simulated by applying a distributed load on the location of the bolt that joins the special bracket to the bogie, as can be seen in the Figure 9.

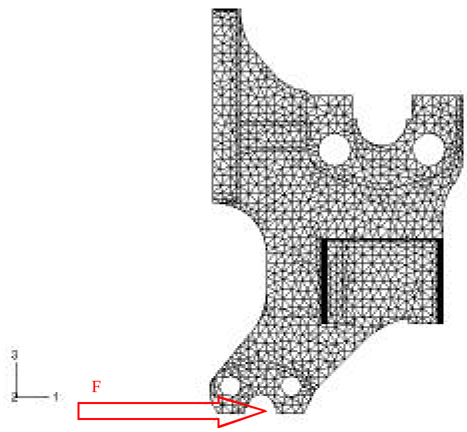

Figure 9. Load applied to the special bracket simulating the acceleration action.

2.6.3 Braking action

Inversely to the previous situation, when the train brakes or reduces its speed the forces coming from the bogie try to push the special bracket to the bolster. This effect was simulated by changing the direction of the load applied to the bolster in the acceleration hypothesis. Figure 10 shows this load case.

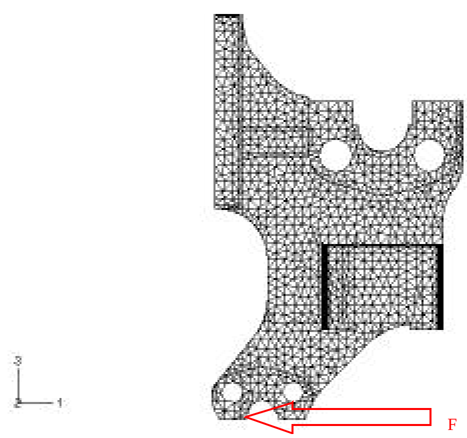

Figure 10. Load applied to the special bracket simulating the braking action.

3. Results

As an example of the results obtained in the substructuring analysis, the figure 11 shows the deformed shape of the bolster for the acceleration load case and the figure 12 shows the deformed shape of the special bracket/superelement for the same load case.

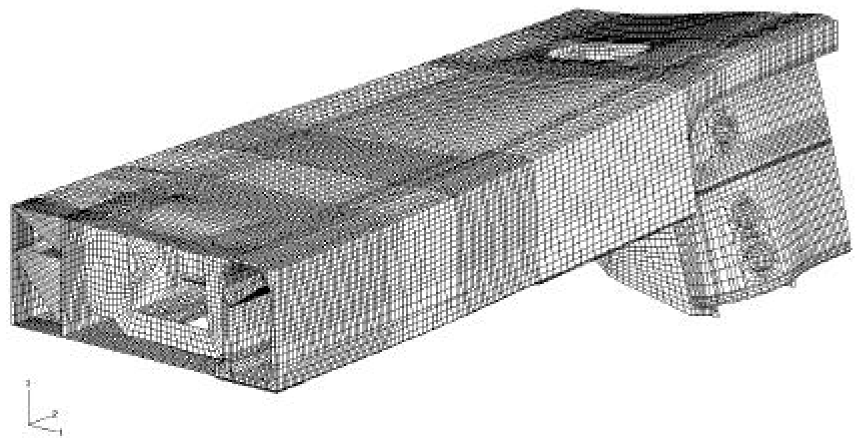

Figure 11. Deformed shape of the bolster for the acceleration load case.

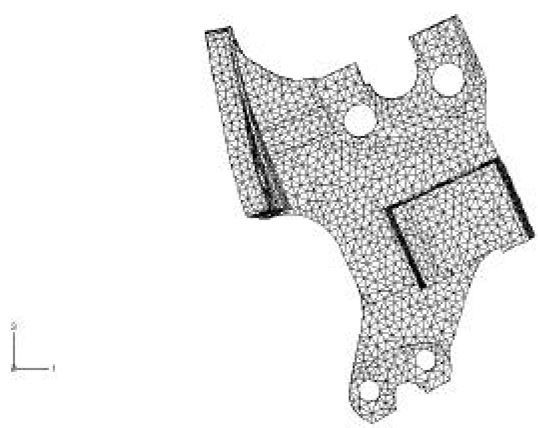

Figure 12. Deformed shape of the special bracket (superelement) for the acceleration load case.

The alternate nature of the loads associated to the acceleration – braking cycles produces fatigue on the weldings of the bolster an so a detailed principal stress analysis in such welding chords was also made attending to the specifications included in the Eurocode 9 Part 2.

4. References

1. ABAQUS Standard version 6.3, ABAQUS User´s Manual, Vols. I, II & III. Hibbit, Karlson & Sorensen, Inc. 2002.
2. Zienkiewicz, O.C. and Taylor, R.L, “The Finite Element Method ” Vols. I & 2, CIMNE, Barcelona, 2000.