Mechatronic analysis of a flexible mechanism using SAMCEF: application to robotics

Michaël Bruyneel and Didier Granville
SAMTECH s.a., Liège Science Park
Rue des Chasseurs-ardennais 8, B4031 Angleur, Belgium
www.samcef.com

Summary:

This paper aims at presenting the solution available in the SAMCEF finite element code for the numerical simulation of the dynamics of flexible mechanisms, including their control.

The development of a virtual prototype in a unique (i.e. fully integrated) SAMCEF environment is described on the specific case of a six-axes robot, which is studied in the frame of the European project RAPOLAC. It is shown how important it is to consider flexibility in the model, and how controllers can be used to impose given movements to the tool. Finally it is briefly explained how optimisation can be applied in this context, both on structural and control aspects.

Keywords:
Virtual prototype, system dynamics, flexible mechanisms, mechatronic analysis, control, robotics, optimization
1 Introduction

The development of simulation tools for the dynamic behaviour of articulated flexible multi-body systems is becoming a challenge for the industrial software providers since the industry is more and more interested in a solution for such problems [1]. In order to accurately analyse an articulated system during its movement, the model must include not only the classical mechanical parts but also the other components which allow and guide the motion, i.e. joints, motors, sensors, actuators and control systems. The resulting mechatronic model can therefore describe the true physical non-linear dynamic behaviour of a machine in working conditions, which interacts dynamically with its controllers [2]. The virtual prototype is a model of a real system, which allows a very deep study of its behaviour, that would be possible on a physical prototype only adopting very complex and expensive instrumentation systems able to measure robot motion and deformation during experiments. The mechatronic model should be used to e.g. reproduce reliably static stiffness tests and tool tip impact tests, to evaluate the dynamic compliance at the tool, to execute specific trajectories and to estimate the machine precision, and to tune the gains of the control devices.

The development of accurate mechatronic models should rely on computational tools able to simulate different levels of fidelity, from the base component up to the full mechanism, including flexibility and possible non-linear characteristics for the components and for the kinematic joints. The SAMCEF finite element code includes all those computational capabilities.

At the end of the Seventies, the design of structural components was carried out in linear elasticity with *SAMCEF Asef* for static analyses and with *SAMCEF Dynam* for modal analyses [3]. Over the Eighties, non-linear material models were made available in SAMCEF and an integrated approach to mechanism analysis in a finite element formulation was proposed [4] in *SAMCEF Mecano*. At the same time several facilities were developed in SAMCEF for sizing and shape optimisation problems [5,6] and the specific optimisation environment *BOSS quattro* was finally developed [7,8]. In the middle of the Nineties, the development of *SAMCEF Field* was started, which is now a fully integrated CAE environment for linear and non-linear structural analyses, for multi-body simulation based on FE methods, for multiphysics problems, and topology optimisation. Over the last 7 years, the optimisation of mechanisms [9], their synthesis from scratch [10] and the solution to mechatronic problems [11,12] were investigated through several European projects and improved those computational tools [13,14,15]. This methodology has been recently extended to wind turbine design [16].

In this paper the development of a virtual prototype in the fully integrated SAMCEF Field environment is described on the specific case of a six-axes robot, which is studied in the frame of the European project RAPOLAC [12]. Through this application it is shown how important it is to consider flexibility in the model, and how controllers can be used to impose given movements to the tool. Finally it is briefly explained how optimisation can be applied in this context, both on structural and control aspects.

2 The RAPOLAC European project

The goal of the RAPOLAC project [12] is to test and simulate the Shaped Metal Deposition process (SMD) able to quickly produce aerospace parts. This original process consists in creating components from the base up in a layer-wise fashion, depositing weld material according to the CAD model, without the need for tooling. A SMD cell is operating at the Advanced Manufacturing Research Centre of Sheffield [12], and the process is managed by two KUKA robots [17], presented in Fig. 1. In this paper the general methodology for developing a mechatronic model with SAMCEF is presented on the KR 16 robot.

![Fig. 1. The DKP 400 and KR16 KUKA robots studied in the project](image-url)
3 The solution procedure available in SAMCEF

SAMCEF is an implicit finite element code. The solution procedure available in SAMCEF for flexible mechanisms was developed at the end of the eighties [4] and is broadly presented in [18]. The main ideas are briefly reported here.

3.1 Finite Element versus classical MBS approaches

The purpose of developing a multi-body approach based on finite elements in [4, 18] was clearly linked to the generality of the solution procedure, the fidelity of the model and therefore to the accuracy of the results. It should be evident (and this will be partly demonstrated in this paper) that using rigid elements at the components and joints levels leads to the definition of a virtual prototype that is definitely not able to reproduce accurately the dynamics of a system. Flexibility is a key issue, which must be modelled at least for two important reasons. Firstly flexibility and inertia produce vibrations, that can influence the precision of a machine and its control strategy. Secondly this is an access to strains and stresses at the material level, both needed to design the structural components.

Including flexibility in the model is typically done based on the finite element formulation, what leads to an increase of the models’ size. In order to decrease the CPU time while keeping (a possible approximation of) the flexibility, the Super Element technique is used. In this case, the deformations are represented by a combination of pre-selected vibration modes of the components for given boundary conditions and the size of the model can remain reasonable.

However using Super Elements for all components limits the level of fidelity that a model can reach. Real life problems are (unfortunately?) non linear and Super Elements can only work under the linear assumptions for the set of eigen-frequencies and for the boundary conditions selected for representing the deformation of the structural components. When this set changes or when the boundary conditions are modified this solution procedure is no longer valid. This is the case when impact tests are studied, i.e. contact must be taken into account and plasticity becomes a design criteria to decide on the damage tolerance of a machine. On top of that, flexibility and non linear behaviours (as friction) should be sometimes modelled at the joints level, since vibrations transit by the joints and can be damped by a too rigid (non real) assembly assumption.

Here a more general solution procedure is presented with SAMCEF, and the full finite elements capabilities of this software can be used to model, with different levels of fidelity for the different components and joints, a complete system subjected to dynamic forces, relying on a very comprehensive elements library including bars, beams, shells, volumes, Super Elements, and flexible joints.

3.2 Formulation and solution of the FE problem

In the finite element method, the system of equations to be solved is given in (1), where \( M \) is the mass matrix, \( \dot{q}, q \) and \( \ddot{q} \) are the accelerations, the velocities and the displacements, respectively, while \( g \) are the forces.

\[
M\ddot{q} = g(\dot{q}, q, t) = g^{\text{external}} - g^{\text{internal}}
\]  

When constraints \( \Phi \) are considered in the problem (e.g. relations between degrees of freedom to make a motion useful to a desired purpose, i.e. kinematic constraints), the problem to be solved is based on the stationnarity of an augmented Lagrangian defined by the kinetic and potential energies and two additional terms related to the constraints including a penalty and the Lagrangian multipliers. It turns that the system of non linear equations to be solved is given in (2), with \( B \) the gradient of the constraints, \( p \) the penalty factor and \( k \) a scaling factor. For the time integration, the Newmark or the HHT schemes are used. The non linear system is solved by the Newton-Raphson method.

\[
\begin{bmatrix}
M + B^T (k\lambda + p\Phi) = g(\dot{q}, q, t) \\
k\Phi(q, t) = 0
\end{bmatrix}
\]

Detailed equations and solution procedures can be found in [3].
The SAMCEF Field environment

The model is developed in a unique (i.e. fully integrated) environment called SAMCEF Field (Fig. 2). The different steps for building the virtual prototype of the robot in SAMCEF Field are:

- importation of the CAD model (here a STEP format, but other standard formats are readable);
- possible modification of the geometry, through standard facilities of shape healing;
- definition of the data assigned to each component based on the geometry, and not on the mesh (material properties - linear or not, behaviour - flexible or rigid, boundary conditions, loads, ...);
- possible importation of Super Elements. SAMCEF Field manages parts: when Super Elements are used, they are considered as parts and are easily imported in the model. Besides the SAMCEF ones, Super Elements of the large commercial general purpose finite element codes can be managed;
- assembly of the moving parts (through hinges in the case of the robot);
- mesh of the mechanical parts;
- definition of the settings for the solution (Newmark, HHT, Newton-Raphson, ...);
- post-processing for the results.

Fig. 2. The SAMCEF Field working environment

4 Models of the 6-axes robot

Different levels of fidelity in the mechanical model are investigated (Fig. 3). The first one is related to a rigid modelling of the structural parts, leading to the study of rigid multi-body systems. In this case, simple rigid bars (shown in the Figure) model the mechanical components, which are reduced to an equivalent mass and the inertia at their gravity centre. In the following levels, the FEA is used to include flexibility in the components. In the Super Element approach some vibration modes are kept and used as a basis for modelling the possible deformations of the body. To avoid the assumptions and limitations of the Super Element technique, the third level is much more general and accurate, since a full finite element approach is considered. Finally a fourth level of fidelity can be defined, when the control system is embarked in the flexible mechanical model, leading to the development of a virtual prototype based on a mechatronic approach.

Such levels of fidelity can also be defined for the modelling of assemblies (joints), in this paper the hinges of the robot (Fig. 4). Up to now only rigid joints have been used, but as they are available in SAMCEF the several possibilities shown in Fig. 4 will be tested and compared in the future.
Prescribed rotations can be defined at the 6 hinges of the robot, and complex simulations of the movement can be easily carried out. However in this paper a simpler movement only impacting the rotation at the hinge linking the base frame and the rotating column is studied (Fig. 5). A rotation from 0 to 45° and back to 0° is applied within 1 second. A time step of 0.01 second is imposed during the solution process. Based on this solicitation, the necessity of considering the flexibility in the model is discussed.
Although SAMCEF is based on a finite element approach it can very effectively (i.e. in a CPU point of view) simulate rigid multi-body systems. When a rigid behaviour is selected for each part of the CAD model, the structural components of the mechanism are replaced by equivalent mass and inertia assigned at nodes located at the components’ gravity centre. The model includes additional nodes at the connections between the parts, and rigid bars are used to link all those nodes (Fig. 3a). The resulting model includes very few degrees of freedom (here around 200) and the resulting system of equations is solved very quickly and easily (about 1 second on a recent PC) by FE solvers developed to manage models including up to millions of degrees of freedom.

4.2 A model including flexibility: the Super Element approach

Flexibility can be managed in the MBS model by using Super Elements as parts of the mechanism, as shown in Fig. 3b. For this purpose, a mesh of the corresponding components must be available. CAD geometries are known to include lots of small edges, small faces and other geometrical defects that should be deleted before the meshing operation. This is illustrated in Fig. 6, where the initial geometry of the link arm is slightly modified before being meshed.

Based on this meshed component a Super Element is defined with SAMCEF. The retained nodes are located at the centres of the hinges, as shown in Fig. 7. The first 20 eigenvalues are kept for the mode component approach used here [19]. In this paper only the link arm is flexible (Fig. 6) while the other parts remain rigid.

The problem of Fig. 5 is solved very quickly, in less than 2 seconds with this Super Element approach.
4.3 A model including flexibility: the full Finite Element approach

As SAMCEF is not a MBS but a finite element code including structural and motion capabilities, using Super Elements is not mandatory at all, and full finite elements models can be considered (Fig 3c). Although more expensive in a CPU point of view than a Super Element (about 3000 degrees of freedom, 100 seconds to solve the problem of Fig. 5, for the given mesh and the imposed time stepping), the full FEM approach is not limited to linear behaviours, and e.g. impact tests and associated damage can therefore be managed (low speed contact and non linear material behaviour). On top of that, no results’ recovery is necessary with a full FEM and the stress levels needed for the components design are directly available.

4.4 Comparison of the results for the 3 first models

Fig. 8 presents the trajectories followed by the tip of the robot’s arm during the movement of Fig. 5 for the three models studied up to now: the one made of rigid parts (on the left), the one including a Super Element (in the middle), and the one defined with a full FEM (on the right). With the rigid models the forward and backward curves are coincident. With the flexible models it is no longer the case. It is clear that the rigid model can not put in light the vibrations that appear when the direction of the movement is changing around 0.5 sec (Fig. 5), since this effect is due to flexibility. It is therefore mandatory to model the flexibility for an accurate modelling of effectively flexible components/system.

---

Fig. 7. Defining the Super Element of the link arm and the resulting model including some flexibility

Fig. 8. Computational models (without CAD representation, unless for the Super Element). Trajectories followed by the tip of the robot’s arm. Stresses for the full FE model
4.5 A mechatronic model of the robot

In the previously presented mechanical models the rotations were prescribed at the hinges and the system response is immediate. In real life machines this is unfortunately not the case since the motion is managed with controllers that include their own dynamics, leading to transient responses before reaching a steady state. An accurate virtual prototype should therefore include such control devices in order to simulate more precisely the machine in working conditions. Here such a device is embarked in the flexible mechanical model. The rotation at the hinge linking the base frame to the rotating column is controlled with a PID available in SAMCEF Field. A sensor and an actuator are located at the hinge defined between the link arm and the rotating column: the sensor measures the actual rotation, and the actuator provides a couple that forces the satisfaction of the set point (Fig. 3d). The gains of the controller are determined with an optimisation approach based on a step of 45° as explained in the next section. Once the gains are obtained for this reference configuration, the dynamics of the virtual prototype can be studied for more complex motions. The results obtained for different set points are reported in Fig. 9. It is clear from those results that the response of the controlled system doesn’t correspond to the imposed value and that the dynamics of the controller strongly influences the behaviour and the accuracy of the virtual prototype.

![Fig. 9. System responses to different solicitations. Comparison of the set point (imposed value) and the process variable (response value)](image)

5 Discussion about the optimisation in mechatronic systems

5.1 Structural optimisation of components

Several optimisation methods are available in the BOSS quattro software [7,8]: genetic algorithms, response surface methods, and gradients based optimisers. For the last ones, derivatives must be computed. Semi-analytical sensitivities for linear and non linear analyses are available in SAMCEF [20] for the static and dynamic cases. Optimal sizing and shape optimisation can be conducted on the structural components [15].

5.2 Tuning the gains of the controller with optimisation techniques

In order to determine the gains of the controller an optimisation problem is defined. In the studied model the control is co-located. The sensor and the actuator are located at the same place, i.e. the hinge between the link arm and the rotating column. The effects of flexibility of the link arm will therefore not influence the control. Based on this, and for CPU time reasons, the optimisation process is carried out with the rigid model. Three design variables are considered, i.e. the gains KP, KI and KD related to the proportional, integral and derivative effects of the PID controller available in SAMCEF Field. Those gains are tuned with respect to a step function of 45°: the robot must perform a rotation of 45° as well as possible and stay in this position over time. The optimal response to this reference step should present no oscillations, no overshoot and a fast rise time (Fig. 10).
The optimisation method described in [21,22] initially developed for structural optimisation proved to be efficient in finding the optimal gains of the controller. It is a gradient based method and the sensitivities are automatically computed by BOSS quattro based on finite differences. The convergence history (i.e. the responses of the system over the iterations) is reported in Fig. 11. The initial configuration is characterized by large oscillations and a non stable response of the system leading to the unceasing alternative rotations of increasing amplitude of the robot around the axis. After 4 iterations, a stable configuration is found, but the rise time doesn't satisfy the restrictions. Although a faster motion is obtained at iteration 10 large oscillations around the set point of 45° still exist. The solution is finally obtained at iteration 39 for the required precision. Note that at the solution $KI$ is equal to zero.

6  Improvements of the model

Several axes in the development of a more accurate virtual prototype of the robot can be addressed. Amongst others: flexibility of all the structural components, modelling of the flexibility at the joints level (Fig. 4), fully controlled movement at the hinges, and finally trajectories monitoring. Besides the PID used in this study, more sophisticated controllers defined with Matlab/Simulink can be imported in SAMCEF Field. This capability will be tested with the robot. The way to formulate the gains optimisation problem should be more deeply studied as well.

---

Fig. 10. The BOSS Quattro optimisation software and the definition of the optimisation problem

Fig. 11. Iteration history for the optimisation of the controller’s gains
7 Summary

This paper has presented the first steps in the development of a mechatronic model of a robot. It was shown that flexibility must be considered to reflect reality. It was explained how controllers can be managed and how they impact the overall dynamics of the system. The developments are carried out in a fully integrated CAE environment called SAMCEF Field. Robotics is an example of application and the discussed methodology should be applied to other kinds of mechanical systems presenting a dynamic behaviour: e.g. high speed machine tools, engines, rotors, vehicles, deployable space systems, wind turbines, etc.

8 Acknowledgements

The work presented in this paper has been carried out in the frame of the RAPOLAC European project. www.RAPOLAC.eu

9 Literature

[9] SYNAMEC European project – Synthesis Tool for Aeronautical Mechanisms Design