

VIRTUAL DESIGN OF A MACHINE TOOL FEED DRIVE SYSTEM

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In zilele noastre, constructorii de mașini-unei nu mai pot irosi timp și resurse construind și testand prototipuri reale ale mașinilor-unei, în locul prototipurilor reale folosind prototipuri virtual. În acest articol este prezentat stadiul actual al proiectării virtuale a mașinilor unele folosind ca studiu de caz lanțul cinematic de avans poziționare. Comportarea statică și dinamică este evaluată pentru obținerea unui model optimizat al lanțului cinematic de avans al mașinii-unei. Folosind metodele specifice proiectării virtuale, inginerii pot reduce timpul necesar proiectării astfel reducându-se durata necesară lansării unui produs nou pe piață.

Today, machine tool manufacturers can no longer afford to consume time and money building and testing real prototypes of the machine tool model, instead they use virtual prototypes. This paper presents the current state of virtual prototyping of a virtual machine tool; the work focuses on the design of the feed drive system. The structural behavior under static and dynamic loads is evaluated in order to obtain an optimized design of the feed drive. The following steps are detailed in the paper: - Importing the 3D model, - Defeaturing of the 3D model and setup of the Finite Element model, - Optimization of the mesh structure. Using Virtual Prototyping techniques, engineers are able to shorten the design time and therefore to shorten the time needed for pushing to market new products.

Keywords: FE model, Mesh, design modeler.

1. Introduction

In the virtual prototype approach one is able to simulate the kinematic, static and dynamic behavior of the machine tool including all aspects of real life exploitation. Using different software packages one is able to simulate even the cutting process. This complex design concept was enabled by the use of high performance computer technology.

Several types of analyses must be performed for a complete study of the machine tool behavior. The results of some simulations can be further used as input for other analyses. For example, from the cutting process analyses we use forces as input for the dynamic finite element (FE) simulation.

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This paper presents the current state of virtual prototyping of a virtual machine tool. The work focuses on the design of the feed drive system. The main aspects of virtual design concept covered by this paper are the structural and dynamic analysis of a machine tool feed drive using the finite element method. The following steps are detailed in the paper: Defeathering of the 3D model and the setup of the FE model.

Unfortunately at the moment there are no integrated software platforms for the virtual design and analysis of the machine tools. As a result, there are many problems that appear when importing models from between different software applications.

Because of the impact of the feed drives imply on the quality of the finite piece the design demands are very high. That is why accurate analyses must be performed in order to assure a very good behavior of the whole machine tool.

2. Computer Aided Design

The first step in virtual prototyping is building the CAD model. The 3D model of the feed drive must be designed in order to be accepted as input by various software suits that will be used for further analyses. Because it is hard to find an integrated environment in which to perform all analyses, engineers must take into consideration software's compatibilities.

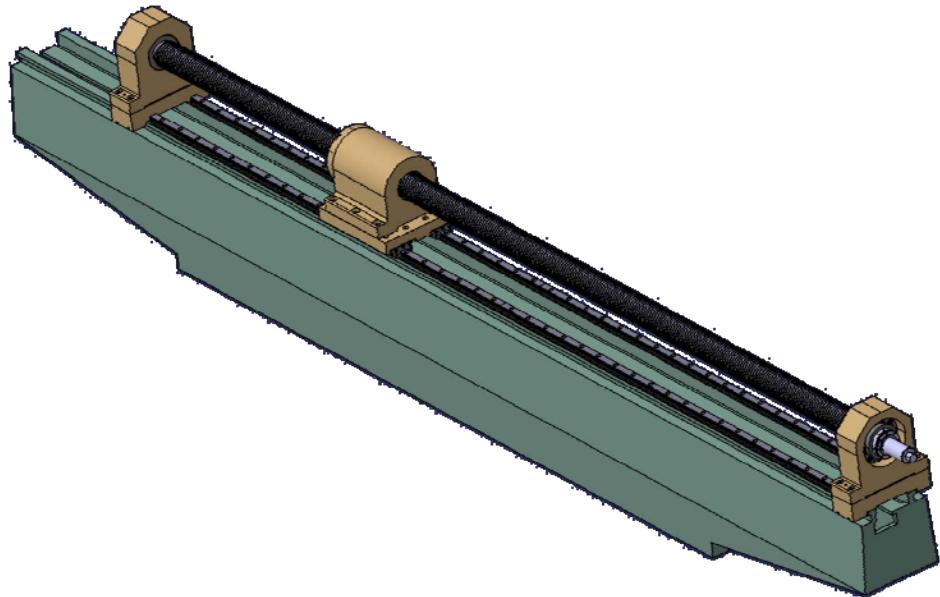


Fig. 1. 3D CAD model of the feed drive

During the design phase simplified simulation models are used to estimate the impact of design parameters over the machine performance. These simplified models are also used in order to improve the time needed for calculation and to eliminate the computing errors. It is well known that a very complex model can generate errors during the FE analysis. Because of this, many features from the 3D model are eliminated. Also, some details that are essential for FE analyses are not so important for a kinematic analyze and must be ignored. A CAD model must be easily redesigned so it is important to use parameters to define all the key elements of the model.

The 3D model of the feed drive was designed using the CATIA V5 CAD software mainly because of its good integration with the ANSYS software which was used for static and dynamic FE analyses. All 3D part were fully parameterized in order to optimize needed parameters in FE analyses.

In order to generate all the contacts between surfaces it's necessary to correctly design the 3D assembly, it is also very useful to check all the clashes and clearances within the CAD environment. Using the information provided by the CAD software, we can set the correct tolerances for the automatic contact generation.

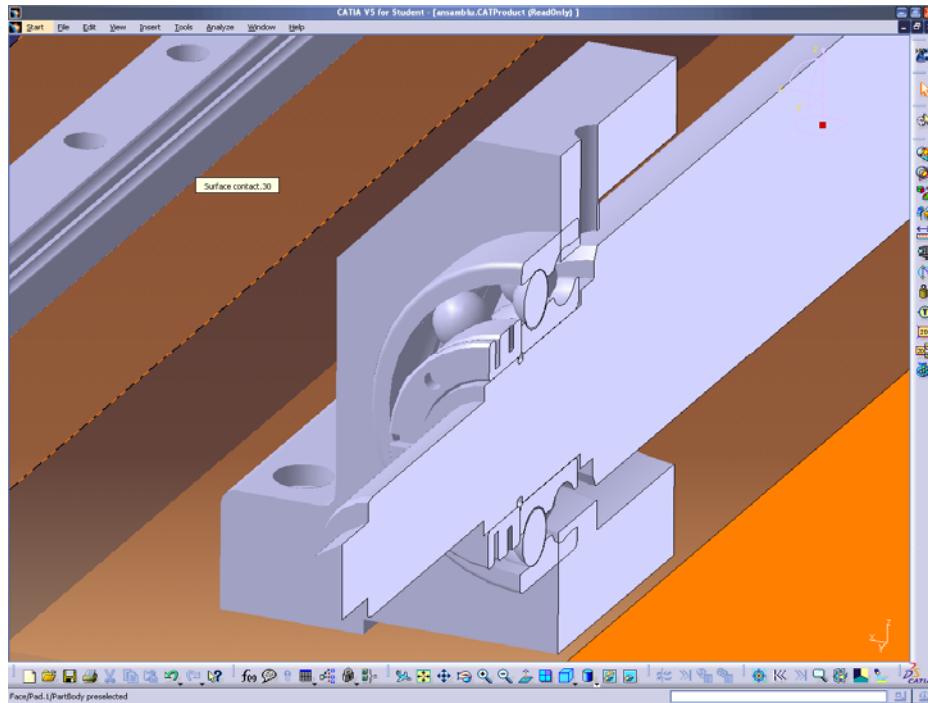


Fig. 2. Section through 3D model

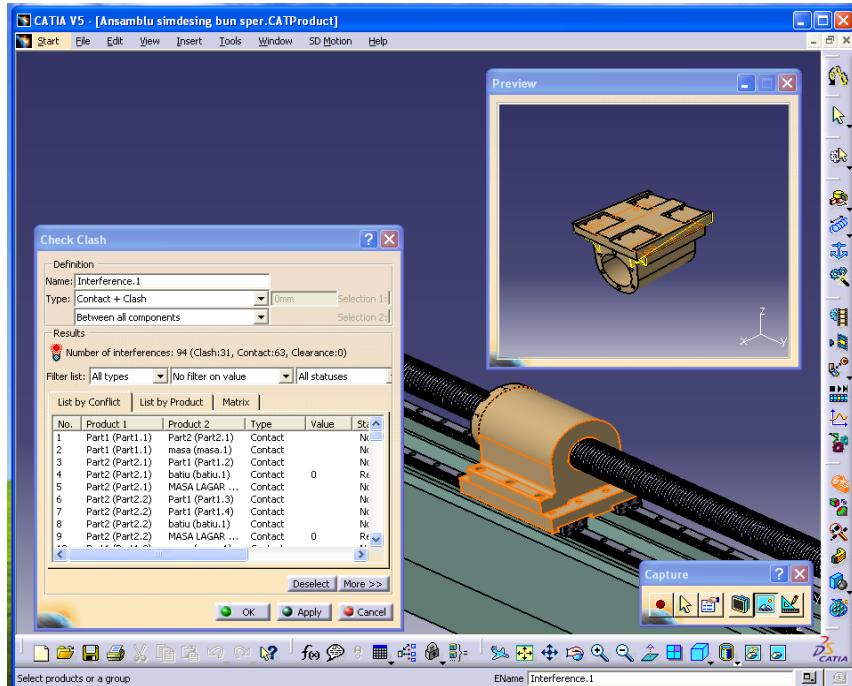


Fig. 3. Space analysis of the 3D model

As a main phase in the design process, the kinematic analysis can be useful because during this simulation all the components move relative to each other respecting the assembly constraints. During this simulation, the mechatronical system can be observed in different stages and it is possible to freeze one position which can be latter imported into the FEM software.

The CATIA software provides some very useful tools for space analysis. In Figure 2 it is an example of a misplaced component. In this case the error is detectable by a visual inspection of the model but sometimes the errors are undetectable by the human eye so the check clash tool must be used (Figure 3). This tool is very important as it calculates all the distances between the assembly components. Those distances can be further used as an input in FEM software in order to generate correct contact surfaces.

Because we need to use the 3D model for further analyses it is a good practice to define also material properties by using predefined model from the CATIA library or by defining new materials. By defining material properties we can also check important aspects of the assembly like volume, mass, moment of inertia, etc.

Some of those information are computed by the FE software. In this way we can check if all the components are imported correctly (material properties, units, volumes, etc.).

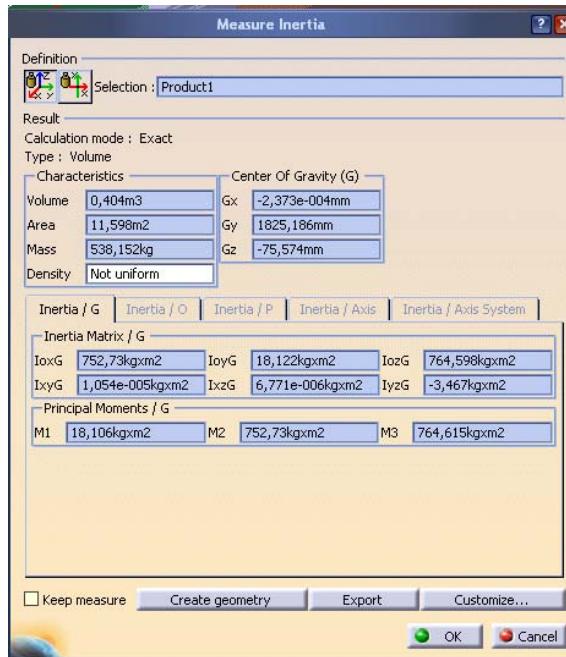


Fig. 4. CATIA inertia measurement tool

3. ANSYS FE analysis

The ANSYS Workbench platform is an environment that offers an efficient and intuitive user interface, superior CAD integration, automatic meshing, and access to model parameters as well as to the functionality available within the ANSYS Mechanical products. Because of the good integration between CATIA V5R15 and ANSYS Workbench we have decided to use this specialized FE software for our static and dynamic analysis.

Because the model was not designed in ANSYS Workbench native CAD system, we must first check if all the model's features are imported before we proceed to further analyses (Fig. 4).

Getting the geometry in Design Modeler, simulation or advanced meshing is now much easier than in almost any other FEM software. The first thing to know about formats is that there are readers and plug-ins. Readers simply translate the CAD format into workbench's internal format. A plug-in actually uses software from the CAD vendor and opens up the geometry in the native format and gives the workbench all the information it needs in native format. Often the geometry provided by readers is referred as dumb and plug-in geometry as smart because this kind of geometry can be transferred back to the CAD files. It's a good practice to import all the parts into the Design Modeler before proceeding to simulation.

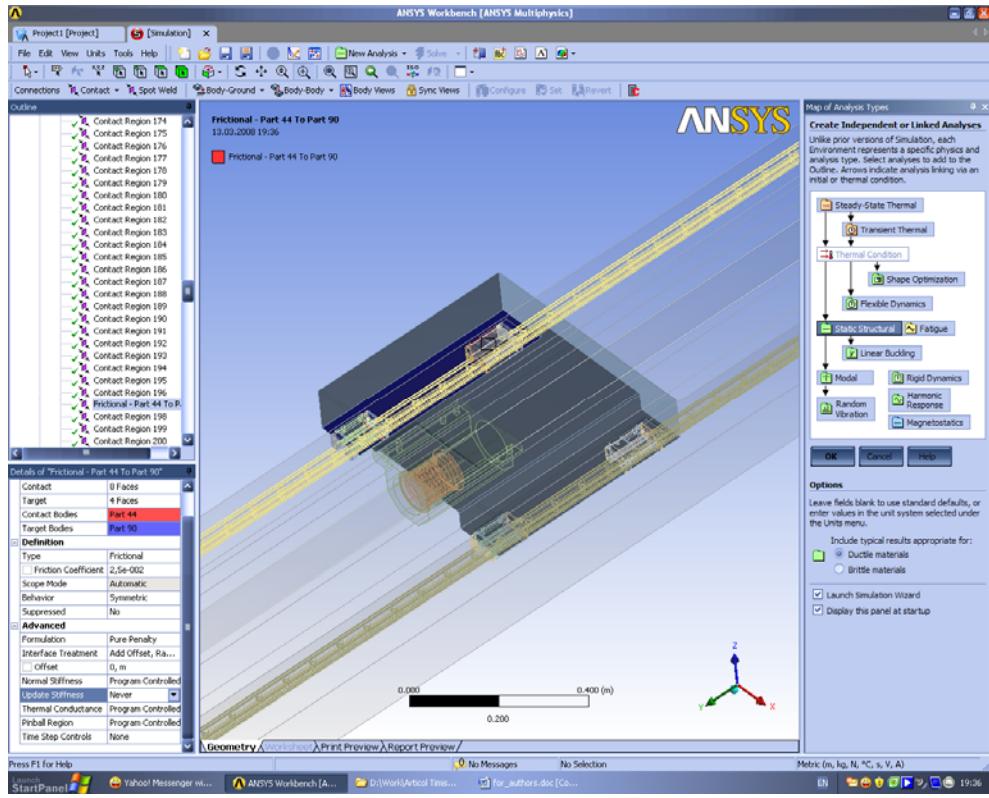


Fig. 5. Contact generation in ANSYS workbench

After a successful import of the model it is very easy to automatically generate all the contacts between components (Fig. 5), setting up the correct parameters for contact creation. It's essential to determine which behavior, formulation, offset behavior to use (Table 1).

There are three basic motions that are described by various behaviors. The contact pair can open, close or slide relative to each other. Depending the behavior type there are different ways that ANSYS enforces these relationships (Table 2)

Table 1

Contact behavior

Name	Gap Open/Close	Sliding allowed
Bonded	NO	NO
Rough	YES	NO, infinite μ
No Separation	NO	YES, $\mu=0$
Frictionless	YES	YES, $\mu=0$
Frictional	YES	YES, if $F_{\text{sliding}} < F_{\text{friction}}$

Table 2

Contact calculation		
Name	Equation Solved	Default Contact Detection
MPC	Generate GE's	Nodes
Pure Penalty	$F=kx$	Gauss
Augmented Lagrange	$F=kx + \lambda$	Gauss
Normal Lagrange	Nodal Pressure DOF	Node

The MPC (Multipoint Constraint) method simply writes constraint equations between the contacting bodies. Pure penalty enforce contact stiffness for a given penetration. The Augmented Lagrange is similar to Pure Penalty, except that it includes a factor for contact pressure to eliminate some chattering effect.

The Normal Lagrange adds a Contact Pressure DOF to the model. In terms of runtime, the formulations shown are listed in increasing run time.

There is a strong connection between contact generation and mesh generation. For a proper meshed structure it is necessary to repair the geometry before any further operations.

The main purpose of the CAD repair tool is to detect and close gaps between neighboring surfaces. Typically, this procedure involve two steps: 1. Build topology – build curves and points which will help to diagnose the model for geometrical problems. If the curves are within a geometric tolerance, they are merged together as one. The curves are then displayed in a specific color to illustrate their connectivity in the surface data, which can be used to determine any gaps or holes in the geometry (Figure 6). 2. Repair any gaps or hole in the topology.

For any two faces (surfaces) that meet at a common edge (curve), there is typically a finite distance between the two edges. By default, a curve is associated with all the edges of each face. This topology of the two surfaces would indicate a gap in the model.

Typically, ANSYS meshers can handle this if the gap is smaller than the proposed element size on the surfaces or curves. Therefore, you would set a tolerance larger than the gap if you are using a large element size.

Table 2

Build Topology Colors

Color	Semnification
Yellow	Single or free edge curves
Red	Double edge curves
Blue	Multiple edge curves
Green	Unattached curves

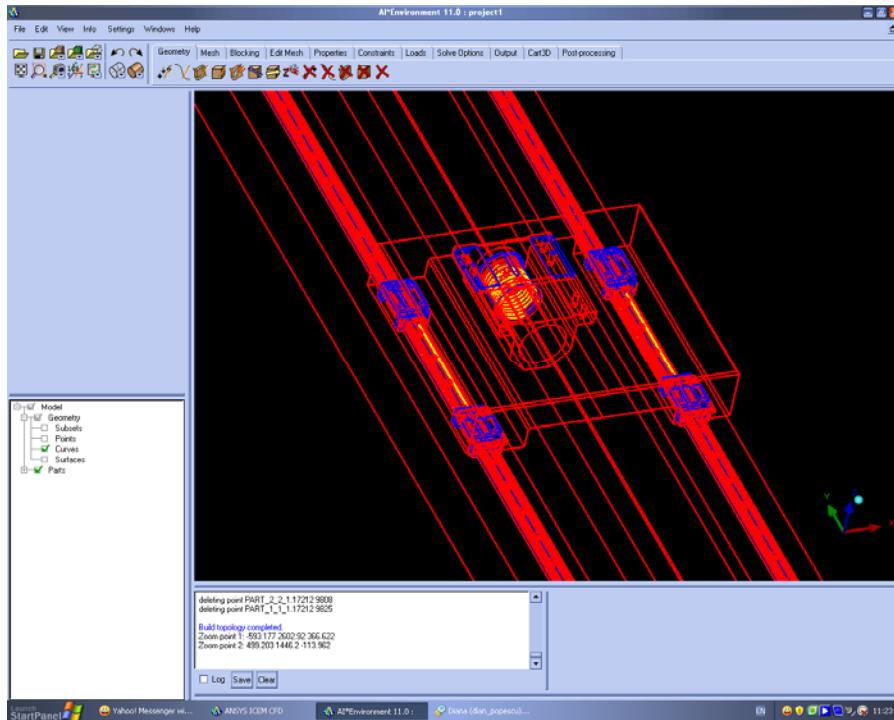


Fig. 6. Mesh quality

A tolerance smaller than the gap would create yellow curves which could be fixed. The recommended tolerance is approximately 1/10th the size of the average mesh size.

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The quality of the mesh needs to be checked before applying loads and constraints. It gives an idea of how close the mesh is to an ideal mesh.

Quality can be measured by the various criteria. For Hexa dominant meshes the quality is calculated as the determinant. The Determinant, more properly defined as the relative determinant, is the ratio of the smallest determinant of the Jacobian matrix divided by the largest determinant of the Jacobian matrix, where each determinant is computed at each node of the element. The Determinant can be found for all linear hexahedral, quadrilateral, and pyramidal elements. A Determinant value of 1 would indicate a perfectly regular mesh element, 0 would indicate an element degenerate in one or more edges, and negative values would indicate inverted elements.

To automatically improve the quality of the mesh elements. Different smoothing algorithms are available depending on which mesh type is loaded.

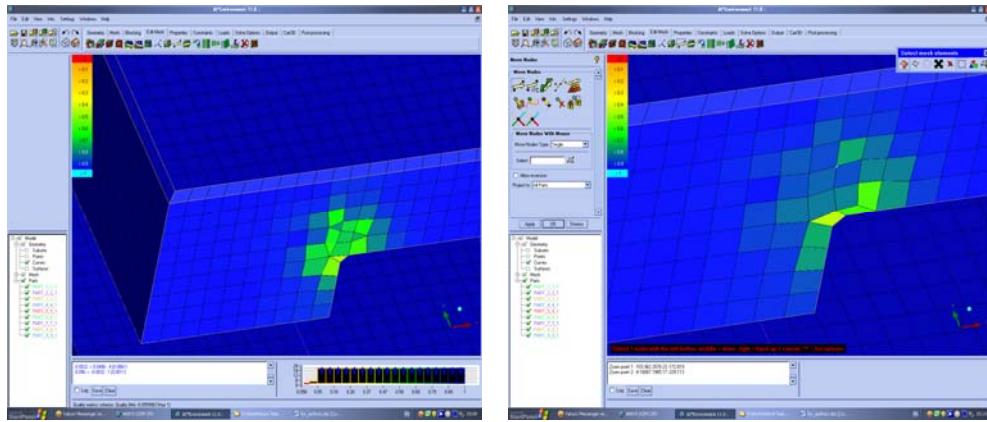


Fig. 7. Manual change mesh quality

Mesh can be smoothed with respect to a particular quality criterion and with a specified number of iterations to achieve a given quality level. A mesh containing tetras, pyramids, prisms and triangular and quad surface elements can be smoothed.

The mesh can be also improved manually by moving nodes. An example is shown in Fig. 7 where some nodes were moved. As a result, we have in the left side of the figure a smoother mesh. This can be checked by visualizing element colors. Those colors change as the form of the elements changes.

6. Conclusions

Using a set of dedicated software, engineers are able to analyze and optimize many aspects of real life usage of machine tool elements without spending money on real prototypes and this could bring important time and money savings.

By using ANSYS Workbench it is possible to optimize the design process by changing one or more of the initial parameters; those parameters are automatically updated into the CATIA 3D CAD model. By analyzing the calculation result in the post-processing program the designers can evaluate the machine properties during the design stage.

Today the main problem in checking structures consists in importing and preprocessing the CAD model. It is well known that the geometry of the model can dramatically change FE results.

Initiated mainly by the automotive and aircraft industry, the development of modern software tools for integrated simulation of products has been enhanced lately. Unfortunately at the moment there are no integrated software platform for the virtual design and analysis of the machine tools.

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