Static and dynamic characterization of new parallel high speed machine tool

*B. C. Bouzgarrou, *P. Ray and *G. Gogu  
**C. Barra and **O. Vidal

(*)Laboratoire de Recherches et Applications en Mécanique Avancée  
Institut Français de Mécanique Avancée et Université Blaise Pascal (Clermont II)  
Campus de Clermont-Ferrand / les Cézeaux BP-265  
63170 Aubière, France.  
(**)PCI-SCEMM, Recherche & Développement  
Rue Copernic, 4230 Saint-Etienne, France.

ABSTRACT. The characterization of a machine tool behavior overall its work space is necessary in the design stage in order to guarantee some minimal performances in term of static and dynamic stiffness. High Speed Machining (HSM) requires high static and dynamic stiffness while minimizing the moving masses. For a machine tool with parallel kinematic architecture, the knowledge of positions for which these performances are minimal is not immediate as for serial machines. The variability of kinematic local properties (distance to singular positions) is coupled with configuration changes of the structure. In this article, we present a finite element analyses of the static and dynamic stiffness of a new HSM machine tool type. This machine have an hybrid kinematic architecture with 3 degrees of mobility: The Z axis is embarked on parallel planar mechanism having 2 degrees of mobility. Stiffnesses of guidance components and of the motors are considered in the model. Performances are estimated overall the work space by analytical cubic forms according to the spindle spatial coordinates. Coefficients of each model are identified by numeric experiment design. Stiffnesses and mode shapes are determined to identify possible topological modifications to be performed.

KEY WORDS. high speed machining, parallel mechanisms, static stiffness, dynamic stiffness, experiment design.
1 Introduction

Important developments have been achieved in the high speed machining (HSM) field concerning machining technology as well as machines’ structures. The use of the parallel kinematics in this field is relatively recent. A generalized parallel manipulator is a mechanism having closed kinematic chain whose end effector is joined to the basis by several independent kinematic chains [MER 97]. When the number of chains that joint the end effector is strictly equal to the degree of mobility of the mechanism, the manipulator is said fully parallel. The parallel kinematics allows to reduce moving masses and to increase displacements speeds. The disposition in parallel of kinematic elements stiffens significantly the manipulator structure. Stiffness of fully parallel manipulator has been studied while considering linkages as springs with constant stiffness [MER 97]. In this case, one demonstrates that stiffness is directly related to local kinematic properties, given by manipulator Jacobian matrix. We can have parallel mechanism carrying another parallel or serial mechanism, in this case global mechanism is said to have hybrid kinematics. Geometric and kinematic models of hybrid manipulators are simpler to establish than for the fully parallel manipulators for the same degree of mobility. Thereafter, their control is easier to program. But stiffness cannot be estimated only according to local kinematic properties. Mechanical components don't act in the same way in all positions.

The most important criteria for dimensioning HSM machines are static and dynamic stiffness. In practice, we try to have them the most elevated possible to reach high performances in term of machining precision and displacement speed and acceleration. The finite elements analysis permits to characterize the static and vibratory behavior of structures. In the first part, we present the approach used in the modeling of the machine. Stiffness of guidance components as well as the one induced by motors, have been considered. In second part, static performances of the structure are analyzed overall the work space using numerical experiment design. Finally, natural frequencies and mode shapes for the minimal stiffness configuration are determined.

2 Modeling

2.1 Description of the machine

Milling centers have generally 3 degrees of mobility (3 axis). The machine that we describe below, Figure 1, is destined to this type of applications. We define kinematic element as the set of component that have the same rigid body motion. The Cartesian milling centers, 3 axis, have 4 kinematic elements, including the basis, related by the 3 prismatic joints. Parallel kinematic machines have more joints than degrees of mobility. The Delta architecture [CLA 91] have 10 kinematic
elements and 15 joints. The use of hybrid kinematics as well as non-fully parallel kinematics permits to reduce mechanism complexity for the same degree of mobility. Indeed, Eclipse\textsuperscript{1} machine, not fully parallel, have 6 degrees of mobility with 10 kinematic elements and 12 joints while 13 kinematic elements and 18 joints are needed for hexapod machines.

The machine that we present has hybrid kinematics: the Z axis is embarked on a plane parallel mechanism that has 2 degrees of mobility. This mechanism performs motions in the XY plane. This disposition presents a reduced number of kinematic elements and joints. We have 5 elements and 6 joints. The elements set is formed by 2 caissons, 2 arms and a spindle carrier. The two caissons are related to the basis by revolute joints. The 2 arms are related by a revolute joint to each other and they are related by prismatic joints to caissons. Finally, the spindle carrier is related by prismatic joint to one arm. The 3 prismatic joints are motorized by the linear motors.

\begin{figure}
\centering
\includegraphics[width=0.8\textwidth]{cad_model.png}
\caption{CAD machine model, PCI\textsuperscript{2} registered model}
\end{figure}

\subsection{Finite elements modeling}

Finite elements analysis of structures gives in general a satisfactory approximation of the real behavior. When analyzing a complex structure, as we present in this

\textsuperscript{1} Eclipse: machine developed by Seoul National University, Korea
\textsuperscript{2} PCI: Process Conception Ingénierie, Saint-Etienne, France.
article, two aspects are to consider. The first consists in finding a compromise between precision and model complexity. The second, is about the application of the method. Indeed, it is not foreseeable to mesh by finite elements guidance components especially when they present some rolling elements. However, taking in account their flexibility is indispensable in our analysis. In the other hand, the analyzed structure is a closed kinematic chain mechanism; therefore it is necessary to respect kinematic constraints and adapt the meshing according to the considered position in the work space. The analysis of the structure has been performed in ANSYS® software, Figure 2. The basis has not been considered.

Figure 2: finite elements model, ANSYS®

2.2.1 Elements definition
The choice of elements is determinant for the validity and the precision of a finite elements calculation. The time of calculation is also an important factor when it is about doing iterations such as for optimization study. Components that we defined can be modeled by shell elements, shell63 [ANS 94]. These elements have 4 nodes with 6 degrees of freedom each. Volume elements, solid45 [ANS 94], have been used for modeling linear guidance rails. They have 8 nodes with 6 degrees of freedom each.

2.2.2 Linear guidance modeling
A prismatic joint allows motion in the guidance direction. However, it presents a flexibility in the other directions. For the studied machine, the linear guidance with
Rollers have been adopted. Their stiffness is given by manufacturer data. We restricted our study to the approached linear behavior around the working zone. The stiffness of the rail sliding block pair was modeled by longitudinal spring elements, *combin14* [ANS 94]. We used 8 elements per pair, 4 elements according to each direction, Figure 3.

![Figure 3: modeling of linear guidance stiffness](image)

### 2.2.3 Bearings modeling

Revolute joints are located between caissons and basis as well as between the two arms. Bearings are used as guidance components. In our analysis, we considered that bearings are perfectly rigid since their stiffness, when preloaded, largely exceeds the other structural elements stiffness. Revolute joints are then modeled by nodal constraint equations compatible with authorized displacement fields.

### 2.2.4 Motors modeling

Prismatic joints allow a mobility in the guidance direction. The static equilibrium of the structure is then assured by forces applied by linear motors. Since we are interested to the dynamic behavior of the machine, we have to consider the controlled structure. Indeed, the applied forces are directly related to the control parameters. In static position, these forces act like springs. Springs stiffness is proportional to the position gain factor of the control loop. To model this behavior, we use elastic elements between caissons and arms. We cannot use in this case longitudinal spring elements because forces are tangent to surfaces. We model the stiffness induced by motors by *matrix27* elements [ANS 94], that are more general that *combin14*, Figure 4. Linear motors apply also magnetic attraction forces between primary and secondary parts that preload the structure. Therefore, it is necessary to verify that the distance between two parts reminds acceptable. These forces are modeled by pressure fields uniformly distributed.
3 Analysis and results

Several analysis types are possible from the finite element model. The most important criteria in machine tool structural design are static and dynamic stiffness. Mechanical resistance of components can be verified in later developments. In this section we present static and modal analysis of the structure. Their characterization overall the work space is useful for optimizing control strategy.

3.1 Static analysis

3.1.1 Stiffness matrix definition

We define the stiffness of the machine by a matrix, $\mathbf{R}$, that maps the displacement vector $[u_x, u_y, u_z, r_x, r_y, r_z]^T$ at the tool center, to machining forces vector, $\mathbf{F} = [F_x, F_y, F_z, M_x, M_y, M_z]^T$, applied at this point. $\mathbf{R}$ is a $6 \times 6$ matrix. We have the relation

$$ \mathbf{F} = \mathbf{R} \mathbf{u} \tag{1} $$

In the following analysis, we disregarded momentum of machining forces applied to the spindle, that depends on the tool length, as well as the rotational displacements. Only linear forces and displacements had been considered. In this case $\mathbf{F} = [F_x, F_y, F_z]^T$, $\mathbf{u} = [u_x, u_y, u_z]^T$ and $\mathbf{R}$ is a $3 \times 3$ matrix. To identify the 9 terms of the matrix $\mathbf{R}$, 3 linearly independent load cases must be studied. By applying a force $F_i$ parallel to $i$ direction, $i = x, y, z$, we can evaluate by finite elements analysis a displacement vector $\mathbf{u} = [u_{ix}, u_{iy}, u_{iz}]^T$. We define compliance terms $s_{ij}$, $j = x, y, z$, by the following relation

$$ s_{ij} = \frac{u_{ij}}{F_i} $$
Assuming that the structure is linear, superposition principle can be applied. Thereafter, for any arbitrary force \( F = [F_x, F_y, F_z]^T \), terms of displacement vector are given by

\[
u_j = F_x s_{ij} + F_y s_{ij} + F_z s_{ij}
\]

Which can be written in a matrix form

\[u = SF\]

\( S \) is a compliance matrix having \( s_{ij} \) as general term. Using equation [1], we obtain the stiffness matrix

\[R = S^{-1}\]

### 3.1.2 Design experiment analysis

The lower stiffness position corresponds to the configuration for which the spindle carrier spreading out is maximal. The knowledge of the corresponding spindle position in the plane is not immediate. To characterize the structure overall the work space, we perform a numeric experiment design. The cost and the precision of such analysis depend on the number of calculation points. We use a full \( 3^n \) factorial experiment design \([BEN \ 94]\). Variables correspond to the \( X, Y \) and \( Z \) spatial coordinates of the spindle, \( n = 3 \). Which yields to 27 experiments for each load case.

We apply three load cases, \( F_x = 3000N, F_y = 3000N \) and \( F_z = -3000N \), in each measurement point \( k, k = 1 \ldots 27 \). Thereafter we get displacements using finite elements calculation. First, we identify compliance matrix \( S_{ij,k} \). \( s_{ij,k} \) terms are force independent. They are approximated overall the work space by cubic regression functions. The use of cubic function is physically consistent if we consider the analogy to beam theory. The adopted regression functions have the following form

\[
s_{ij,k} = a_{1,ij} x_k^3 + a_{2,ij} y_k^3 + a_{3,ij} z_k^3 + a_{4,ij} x_k^2 y_k + a_{5,ij} y_k^2 z_k + a_{6,ij} z_k^2 x_k + a_{7,ij} x_k y_k^2 + a_{8,ij} y_k z_k^2 + a_{9,ij} z_k x_k^2 + a_{10,ij}
\]
\((x_k, y_k, z_k)\) are the coordinates of \(k^{th}\) measurement point. \(a_{i,j}, i = 1...10\), are the regression functions coefficients. They are identified in least square meaning. Let \(x_{ij} = [a_{1,ij}, ..., a_{10,ij}]^T\) be the coefficients vector and \(b_{ij} = [u_{ij}, ..., u_{27,ij}]^T\) the vector of displacements in measurement points. In optimal case, coefficients verify the relation

\[
Ax_{ij} = b_{ij}
\]  

[7]

where \(A\) is a \(27 \times 10\) matrix given by

\[
A = \begin{bmatrix}
x_1^3 & y_1^3 & \cdots & z_1x_1^2 & 1 \\
\vdots & \vdots & \ddots & \vdots & \vdots \\
x_k^3 & y_k^3 & \cdots & z_kx_k^2 & 1 \\
\vdots & \vdots & \ddots & \vdots & \vdots \\
x_{27}^3 & y_{27}^3 & \cdots & z_{27}x_{27}^2 & 1
\end{bmatrix}
\]  

[8]

We can remark that \(A\) is identical for all compliance terms. Finally, coefficients are determined by using \(A\) pseudo-inverse matrix [9].

\[
x_{ij} = \left(A^T A\right)^{-1} A^T b_{ij}
\]  

[9]

The use of numeric experiment design permits us to estimate stiffness matrix terms overall the work space by response surfaces. The origin of the reference coordinate system is taken in the center of the work space. Graphics, Figure 6, are projections of response surfaces over several planes, \(Z\) constant. Presented values are dimensionless. They are divided by reference stiffness value. It is to notice that stiffness according to the \(X\) direction is globally lower because of the rails disposition in the spindle carrier. That brings us to propose a modification of rails disposition to equilibrate their contribution in term of structural stiffness. We present, Figure 5, static deformed shapes under two different load cases. What shows that the spindle carrier is the more deformed element.
Figure 5: deformed shapes

Figure 6: projected stiffness response surfaces
3.2 Modal analysis

Modal analysis is performed for minimal static stiffness position. We used reduced method for modal extraction. This method permits to reduce the size of the eigen problem. The studied problem has a large size, more than 8000 elements, which corresponds to more than 40000 degrees of freedom (DOFs). In such case, Guyan reduction technique is strongly recommended [GEN 95] [GMU 97]. The quality of the solution depends on the choice of master DOFs. We used the automatic procedure proposed in ANSYS. It consists in taking master DOFs that correspond to the smallest values of the quotient $k_{ii} / m_{ii}$, where $k_{ii}$ and $m_{ii}$ are the diagonal terms of structural stiffness and mass matrices. 300 master DOFs have been retained.

Mode shapes analysis, show that second and third modes are mechanism modes (rigid). The corresponding natural frequencies can be increased by adopting greater position gain factor. The first structural mode corresponds to a deflection about $X$ axis. To increase the corresponding natural frequency, we have to stiffen caissons. The other natural frequencies are greater than $100Hz$. 
Figure 7: 6 first mode shapes
4 Conclusion

In this article we presented a static and modal analysis of a new type of machine tool using finite elements method. Guidance element stiffness as well as the one induced by motors have been considered. Results that we obtain allow to understand the machine behavior and identify topological modification to be performed. In order to refine models relative to guidance elements, we foresee to analyze them by sub-structuring technique in future developments. Modal characterization of the structure overall the work space will also performed to optimize the control strategy.

References:


