THE ANNALS OF “DUNAREA DE JOS” UNIVERSITY OF GALATI.
FASCICLE IX. METALLURGY AND MATERIALS SCIENCE
N^{0}. 4 – 2011, ISSN 1453 – 083X

A NEW CONFIGURATION FOR CAST BEARING SUPPORT FROM THE WORK ROLL AS PART OF A THICK SHEET ROLLING MILL

Stefan DRAGOMIR, Georgeta DRAGOMIR, Marian BORDEI
“Dunărea de Jos” University of Galaţi
email: Stefan.Dragomir@ugal.ro

ABSTRACT

In this work is made an optimization of configuration for casting bearing support of backup rolls, with Finite Element Method. This program is used for configuration of parts or subassemblies realized from cast iron. The deformation and displacement of bearing support are configured by using the “Finite Elements Method”. In this study is shown too, an approval on the behavior of bearing support material (hardness, resistance and elongation).

KEYWORDS: backup roll, bearing support, thick sheet, shape, hydraulic installation

1. Introduction

The optimization of configuration for casting bearing support of backup rolls is made because the old configuration has some damage like: in the zone of corner and welded joint are produced – after a short functioning – some cracks because of the premature fatigue of material.

For the first time it is the bearing structure is checked and we take account of the tensions that deformed this part.

At the next stage, for obtaining the optimal configuration it is necessary to consider the force of lamination, other stresses, their direction, the masses which loaded the bearing and the type of bearing which was chosen.

In the FEM, (FINITE ELEMENT METHOD) the structural system –a back-up roll body-is modeled by a set of appropriate **finite elements** interconnected at points called nodes. Elements may have physical properties such as thickness, coefficient of thermal expansion, d, Young's modulus, shear modulus. Some common element types are listed below:

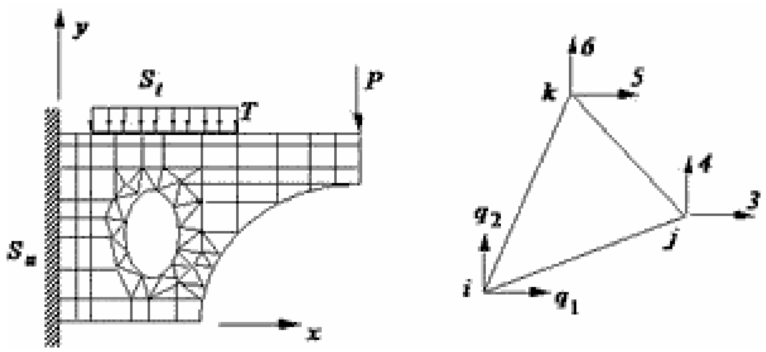


Fig.1. *Mesh (shape with triangular and rectangular elements) for the upper of back up work roll.*

Straight or curved one-dimensional elements are endowed with physical properties such as axial, bending, and torsion stiffness’. This type of elements is suitable for modeling cables, braces, trusses, beams, stiffeners, grids and frames. Straight elements usually have two nodes, one at each end, while curved elements will need at least three nodes including the end-nodes. The elements are positioned at the central axis of the actual members.

Two-dimensional elements are used for membrane action (plane stress, plane strain) and/or bending action (plates and shells).

They may have a variety of shapes such as flat or curved triangles and quadrilaterals. Nodes are usually placed at the element corners and, if needed for higher accuracy, additional nodes can be placed along the element edges or even inside the element.

Three-dimensional elements for modeling 3-D solids such as machine components, or solid masses are required to use different element shapes that can include tetrahedral and hexahedral forms.

Nodes are placed at the vertexes and possibly in the element faces or within the element.

The elements are interconnected only at the exterior nodes, and altogether they should cover the entire domain as accurately as possible.

Nodes will have nodal (vector) displacements or degrees of freedom which may include translations, rotations, and for special applications, higher order derivatives of displacements.

When the nodes displace, they will drag the elements along in a certain manner dictated by the element formulation.

In other words, displacements of any points in the element will be interpolated from the nodal displacements, and these are individual elements.

This is the crucial step where we will need displacement functions written only for the main reason to approximate the nature of the solution.

For obtain a good simulation, it is important to consider the next rules:

-symmetry or anti-symmetry conditions are exploited in order to reduce the size of the domain;

-displacement of finite elements, including any required discontinuity, is ensured at the nodes, and preferably, along the element edges as well, particularly when adjacent elements are of different types, material or thickness;

- compatibility of displacements of many nodes can usually be imposed via constraint relations.

When such a feature is not available in the software package, a physical model that imposes the constraints may be used instead;

-elements' behaviors capture as the dominant actions of the actual system, both locally and globally.

-the element mesh is sufficiently fine in order to have acceptable accuracy. To assess accuracy, the mesh is refined until the important results show little change. For higher accuracy, the elements should be as close to unity as possible and smaller elements are used over the parts of higher stress gradient;

-proper support constraints are imposed with special attention paid to nodes on symmetry axes.

While the theory of FEM can be presented in different perspectives or emphases, its development for structural analysis follows the more traditional approach via the virtual work principle or the minimum total potential energy principle.

The virtual work principle approach is more general as it is applicable to both linear and non-linear material behaviors.

The virtual internal work in the right-hand-side of the above equation may be found by summing the virtual work in the analyzed structure.

The principle of virtual displacement for structural system has the expression - equation (Hook) in the subsequent sections:

$$\sigma = E \cdot з \qquad (1)$$

$$R = Kr + R^0 \qquad (2)$$

where:

R - vector of nodal forces, representing external forces applied to the system's nodes.

r - vector of system's nodal displacements, which will, by interpolation, yield displacements at any point of the finite element mesh.

R^0 – is vector of equivalent nodal forces, representing all external effects other than the nodal forces which are already included in the preceding nodal force vector **R**.

These external effects may include distributed or concentrated surface forces, body forces, thermal effects, initial stresses and strains.

K= system stiffness matrix, which will be established by *assembling* the *elements' stiffness matrices*: K^e.

Once the supports' constraints are accounted for, the nodal displacements are found by solving the system of linear equations (2), symbolically:

$$R = k^{-1} (R - R^0) \qquad (3)$$

Subsequently, the strains and stresses in individual elements may be found as follows:

$$з = B.q \qquad (4)$$

$$\sigma = E \cdot (з - з^0) + \sigma^0 \quad (5)$$

where:

q-vector of element's nodal displacements--a subset of the system displacement vector **r** that pertains to the element under consideration.

B- strain-displacement matrix that transforms nodal displacements **q** to strains at any point in the element.

K- system stiffness matrix, which will be established by *assembling* the *elements' stiffness matrices*: K^e.

Once the supports' constraints are accounted for, the nodal displacements are found by solving the system of linear strains and stresses in individual elements may be found by using equation (4).

E- elasticity matrix that transforms effective strains to stresses at any point in the element and 3^0 is vector of initial strains in the element.

The strain-displacement matrix has the possibility to transforms nodal displacements **q** to strains at any point in the element.

σ^0= vector of initial stresses in the element.

We can apply the virtual work equation to the system and we can establish the element matrices **B** and K^e. The computer program can assembly the system matrices $\mathbf{R^0}$ and K. Other matrices Parameters such as $з^0$, σ^0, **R** and **E** can be directly set up from data input.

2. Interpolation and shape functions

If we consider that q is the vector of nodal displacements of a typical element. The displacements at any point of the element may be found by interpolation functions such as:

$$U=\mathbf{N}\cdot q \tag{6a}$$

where:

u= vector of displacements at any point {x, y, z} of the element.

N= matrix of *shape function* serving as interpolation functions.

Equation (6) gives rise to other quantities of great interest:

Virtual displacements consistent with virtual nodal displacements:

$$\delta u = \mathbf{N}.\delta q \tag{6b}$$

Strains in the elements:

$$з=\mathbf{D}.u \tag{7}$$

where: **D** is matrix of differential operators that convert displacements to strains using linear elasticity theory. Equation (7) shows that matrix **B** in (4) is:

$$\mathbf{B}=\mathbf{D.N} \tag{8}$$

Virtual strains consistent with element's virtual nodal displacements:

$$\delta з= \mathbf{B}.\delta q \tag{9}$$

The shape of the bearing support is shown in Figure 2.

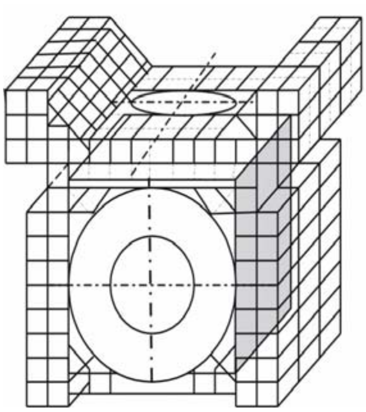


Fig.2. *The configuration of support bearing with finite elements.*

For determining the tensions which loaded the bearing support, the following stages must be included:

-the determination of the exterior and interior tensions applied on this part;

-the measurement of torques, forces and displacement;

-the configuration of the bearing support model and the Finite Elements Method should be applied on this shape;

-the calculus of torques, forces and displacement made by the program and the verification of this structure.

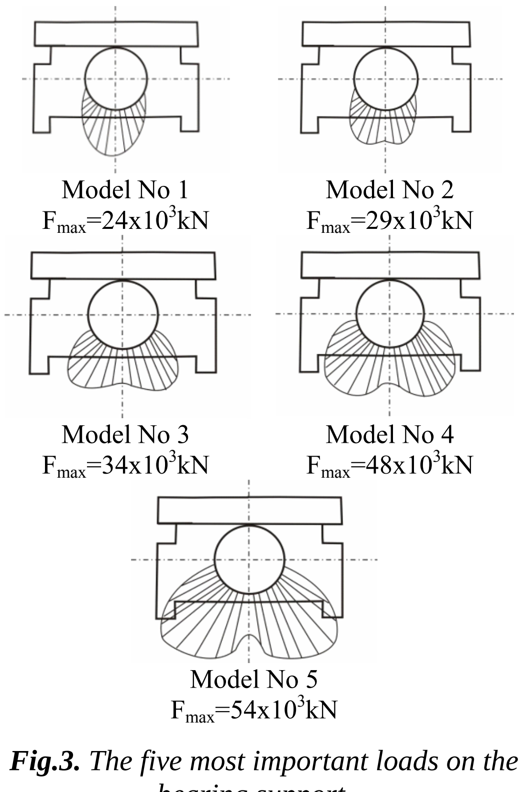
3. Experiments and results

In the experimental phases are recorded the maximum load of $54x10^3$ kN and the displacement of 0.002 mm.

Based on this registered measurement is made a computer simulation.

There were selected five representative models from 476 different experimental measurement and is made a modal analysis by using the IDEAS-Program. The regression-active model is shown in Fig. 3.

The IDEAS program establishes the types of finites elements, their number, arranges the mesh on the shape and shows in dark colour the most tensioned zones and in light colour the less tensioned zones.



Model No 1
$F_{max}=24x10^3kN$

Model No 2
$F_{max}=29x10^3kN$

Model No 3
$F_{max}=34x10^3kN$

Model No 4
$F_{max}=48x10^3kN$

Model No 5
$F_{max}=54x10^3kN$

Fig.3. *The five most important loads on the bearing support.*

After the verification of the resistance of this part, it is chosen the optimal shape for the bearing support.

The ideal shape of the cast bearing support was chosen by following the line of model number 4 (Fig. 3) because it supports the maximum of tensions and deformations: $F_{max} = 48x10^3$ kN and the displacement 0.0187 mm.

3. Material

The material choice for the bearing support was nodular graphite irons Fgn700-2. This material supports (after the experiment) the maximum force of $48x10^3$kN and the less deformation in work.

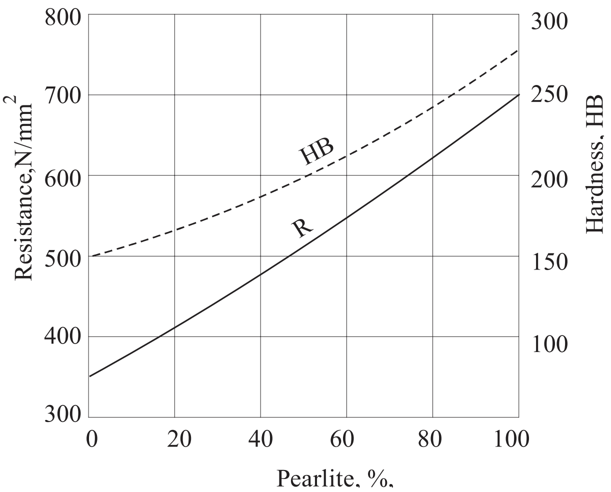


Fig.4. *The characteristics of hardness resistance and elongation for the material Fgn 700-2.*

The growing of pearlite inside the structure determines the increase of cast bearing resistance (R) and the hardness (HB) like in Figure 4.

By using this material is registered the low value for the deformation of the cast bearing support. The equipment used was crucible induction furnace. Like separation plans was chosen the middle of the bearing support and the cast is made in the middle of the bearing shape.

4. Conclusion and result

The modern proceeding for stress and tensions analysis is important for obtaining an optimal shape and a new material for the cast bearing support.

The support model takes account of the tensions, the torques and the dynamics coupling in the functioning time.

It is necessary to chosen the rationally cinematic scheme with a real number of masses and the gap between the bearing and its support.

This part is made from Fgn 700-2 and we chosen iron with nodular graphite because its support the greatest effort at the less deformation.

It must consider the less deformations because in functioning time, a great deformation of bearing support determines the variations of the thick sheet rolling.

References

[1]. Dragomir S. - *Utilaje, agregate, echipamente pentru fabricarea produselor carbonice,* Ed. Universitatii „Dunarea de Jos".

[2]. Dragomir St., Tudor B., *Research about the actionnary system at the continnous discharge heating furnace at ISPAT-SIDEX S.A.* Conferinţa Internaţională de tribologie. Galaţi. ROTRIB'03, PAG.167, 2003,

[3]. Dragomir St., *Monitoring of iron sheet deformation in the rolling mill process by using CVC system, published at*: the-XII International Metallurgy Materials Congress, Istambul, Turkie, pag.843 – 847, 2005.

[4]. Cao, J.G., Qi, J.B., Zhang, J., et al. - *Backup roll contour for edge drop control technology in tandem cold rolling mill.* Journal of Central South University (Science and Technology), Vol.39, No. 5: 1011-1016, 2008.

[5]. Cao, J.G., Mao, N., Zhang, J., et al. - *Study on shifting mathematical model of edge drop control in tandem cold rolling mill.* Iron and Steel, Vol.43, No.8: 57-60, 2008

[6]. Campas, J.J., Terreaux, S., Roches, L.V D, et al. - *New on-line gage for edge drop measurement and effect of tapered work rolls.* Iron and Steel Engineer, Vol.72, No.12:27-32, 1995.

[7]. Wang, J.S., Zhao, Q.L., Jiao, Z.J., et al. - *The research on the principle and application for T-WRS&C cold mill.* Heavy Machinery, No.6:8-11, 2001.