FEM modelling of membrane structure in human hernia repair

Izabela Lubowiecka¹ and Krzysztof Woźnica²

¹Department of Structural Mechanics and Bridge Structures, Gdansk University of Technology
ul. Narutowicza 11/12, 80-233 Gdansk, Poland
e-mail: lubow@pg.gda.pl
and AIMEN Technology Centre, Porrino, Spain

²Ecole Nationale Supérieure d’Ingénieurs, Laboratoire PRISME
88 Boulevard lahitolle, 18020 Bourges Cedex, France
e-mail: woznica@ensi-bourges.fr

Abstract

The paper deals with a Finite Element modelling of human fascia-implant system. The synthetics mesh implanted in the abdomen during the hernia repair surgery is modelled as a membrane structure. Two types of boundary conditions analysed here reflect the two situation in hernia repair and the reaction force in the tissue-mesh connection is calculated in both cases. The material parameters of the mesh are identified in laboratory tests. The system undergoes the internal abdominal pressure appearing within the postoperative cough, that is the most essential load causing the connection failure and the hernia recurrence.

Keywords: biomechanics, membrane, hernia, finite element modelling

1. Introduction

The human ventral hernia is a leak in the musculo-fascial system of the front abdominal wall caused by a fascia defect. Even if the ventral hernia repair surgery is a well recognized matter, mechanical properties of the tissue-implant system are not known so the implantation of the repairing mesh is quite intuitive and, recurrences of the illness happen (see e.g., Ref. [1,5]). Neither the number of the tacks required for holding the implanted mesh correctly is known nor their optimal position on the mesh surface. It is obvious that the tacks can potentially damage human nerves and blood vessels, for that reason their number should be reduced to a minimum. The medical knowledge itself is not sufficient to solve this complex problem therefore it becomes interdisciplinary, bringing together surgeons and engineers.

In this contribution the authors propose two mechanical models of tissue-implant system that differ the issue if boundary conditions. First model with the stiff support in the places of joints refers to the case of the stiff hernia orifice while the second one with elastic supports allows one to take into consideration the cooperation of both system components, tissue and implant [6]. In both cases the crucial result is the reaction force in the connection points that should not exceed the failure load identified in Ref. [7].

2. Material properties of the implant

The experimental tests were carried out on the Proceed® mesh samples undergoing tension in the testing machine Zwick Roel 020 Ref. [3]. The mesh material were identified as orthotropic of the mass density = 256.55 kg/m³ with elastic moduli $E_1 = 83.98e6$ Pa and $E_2 = 12.367e6$ Pa. The Poisson’s coefficient estimated as $\nu_{12} = 0.28$. The orthogonal direction 1 and 2 refers to the mesh structure. The material parameters have to fulfill Eqn. (1) and Eqn. (2) Ref. [2].

$$E_1 \nu_{12} = E_2 \nu_{21}$$

$$\frac{1}{E_1} (\sigma_1 - \nu_{12} \sigma_2), \quad \frac{1}{E_2} (\sigma_2 - \nu_{12} \sigma_1), \quad \gamma_{12} = \tau_{12}/G_{12}$$

3. Numerical models of hernia and the finite element analysis

The implant was modelled as an orthotropic membrane structure. In this approach the hernia is considered as a stiff, circular orifice in a human fascia, which refers to a practical clinical case. The implant is fixed to the fascia at 10 points by tacks in a semi-circular order. The assumed membrane radius is equal to $r = 0.05$ m and the thickness $t = 0.6$ mm.

The nonlinear static analysis has been performed by means of MSC.Marc system. A 4-node finite membrane elements of type 18 (MSC.Marc) containing 3 translational degrees of freedom in each node were applied.

3.1. Model 1 with stiff supports

First, the stiff support in the places of joints were assumed and taken to the analysis (Fig. 1).
These boundary conditions refer to a stiff hernia edge. In the model, the 3 orthogonal displacements are blocked in the points of joints with additional vertical support around the membrane not shown in the figure. In this case, the junction force $R_1 = 5 \text{ N}$ and the maximum deflection $dz_1 = 9.4 \times 10^{-3} \text{ m}$ (Fig. 2) appear.

### 3.2. Model 2 with elastic supports

Next, the model 2, containing the spring supports was developed. This model is more appropriate in the case of hernia where we are able to assume that the tissue has an influence to the membrane behaviour.

In this model (Fig. 3), the springs are provided in the plane of the membrane as well as perpendicular to its reference surface on the edge around the structure. The vertical springs are shown in Fig. (4).

The stiffness of springs was estimated as the stiffness of the system of fascia, muscles and other organs in the abdomen on basis of the laboratory tests Ref. [4]. In this model, the junction force $R_2 = 7.8 \text{ N}$ and the maximum deflection $dz_2 = 1.79 \times 10^{-2} \text{ m}$ (Fig. 4).

### 4. Conclusions

The authors developed a membrane finite element model of the mesh implanted in human body to repair the ventral hernia with a stiff orifice.

The authors proposed two variants of finite element models with the stiff and elastic supports. Both models refer to rigid hernia orifice. Model 1 reveals lower junction force and also lower deflection in the centre of the membrane. In model 2, a higher value of the reaction force appears and the membrane deflects more. These models can be used to estimate the necessary joints number in different situation for laparoscopic surgery.

### References


### Acknowledgements

This study is partially supported by the EU, as part of the Innovative Economy Operational Programme (contract No UDA-POIG.01.03.01-22-086/08-00) and by Bourses du Gouvernement Français. This support is kindly acknowledged. Computations were done in TASK Computer Science Centre, Gdańsk, Poland.