Available online at www.sciencedirect.com

ScienceDirect

Clinical Biomechanics 23 (2008) 881–885

CLINICAL BIOMECHANICS

www.elsevier.com/locate/clinbiomech

Preliminary analysis of the forces on the thoracic cage of patients with pectus excavatum after the Nuss procedure

Pei Yeh Chang [a,*], Zhen-Yu Hsu [b], Da-Pan Chen [b], Jin-Yao Lai [a], Chao-Jan Wang [c]

[a] Department of Pediatric Surgery, Chang Gung Memorial Hospital, Chang Gung University, College of Medicine, Taoyuan, Taiwan
[b] Department of Mechanical Engineering, Chiao-Tung University, Hsinchu, Taiwan
[c] Department of Radiology, Chang Gung Memorial Hospital, Chang Gung University, College of Medicine, Taoyuan, Taiwan

Received 2 August 2007; accepted 19 February 2008

Abstract

Background. The Nuss procedure corrects pectus excavatum using a pre-bent bar that generates stress on the chest wall. To investigate the biomechanical effects after the Nuss procedure, we designed a three-dimensional finite element analysis model to analyze the distribution of stress and strain induced in the chest wall.

Methods. Three patients with pectus excavatum aged 8, 7, and 7 years, were enrolled in this study. The greatest upward displacements of their sternums after the operation were measured from computed tomography images and chest X-ray films. Based on these displacements, we constructed three finite element analysis models for analyzing biomechanical changes in the thoracic cage after the Nuss procedure.

Findings. The simulation results indicated that greatest strain occurred at the third through seventh cartilages, especially where they join the sternum and ribs. A high bilateral stress distribution was also found over the backs of the third to the seventh ribs near the vertebral column.

Interpretation. The stress and strain induced by the Nuss procedure can be analyzed using our finite element analysis model. Although the stress and strain may have some influence on chest and spine development, a more detailed finite element analysis model is recommended for future study to improve the accuracy of our simulation results.

Keywords: Finite element analysis; Pectus excavatum; Nuss procedure; Biomechanical simulation; Stress and strain

1. Introduction

Pectus excavatum (PE) is a common chest wall malformation (Fonkalsrud, 2003) thought to be caused by the excessive growth of the costal cartilage, which produces a concave anterior chest wall (Länsman et al., 2002). In 1998, Nuss introduced a minimally invasive technique for repairing PE (Nuss et al., 1998). In this procedure, the depression of the sternum is corrected by inserting a metal bar (a Nuss bar) under the sternum without removing the costal cartilage. Until recently, most research efforts have focused on improving the Nuss procedure (Nuss et al., 1998; Croitoru et al., 2002; Park et al., 2004b) and preventing complications (Hebra et al., 2000; Park et al., 2004a; Croitoru et al., 2005). However, little mention has been made about analyzing the biomechanical effects on the chest wall of patients with PE who have undergone the Nuss procedure. Because investigating the influence of the stress generated on the chest wall of these patients is extremely difficult, if not impossible, under clinical conditions, we developed a finite element analysis (FEA) model specifically to analyze the stress and the strain distributions induced in the chest wall by a Nuss procedure.

* Corresponding author.
E-mail address: pyjchang@cgmh.org.tw (P.Y. Chang).

0268-0033/$ - see front matter © 2008 Elsevier Ltd. All rights reserved.
doi:10.1016/j.clinbiomech.2008.02.010

2. Methods

2.1. Patient-specific finite element models

The Nuss procedure is an evident biomechanical manipulation of the chest wall caused by the insertion of a metal bar. From our clinical observations, the deformation after the Nuss procedure is different for each patient with PE. FEA models must be generated individually to ensure the possible application of simulated results. It was not our intention to construct a general model of the biomechanical effects for application to all patients.

We carefully chose three symmetric types of patients with PE (Park et al., 2004b) for our study to eliminate the factors due to asymmetric configuration of the chest wall. The patients were 8, 7, and 7 years old, and their pectus indexes were 5.3, 4.7, and 5.2, respectively (Table 1). The first and the third patients were boys and the second was a girl. All of these patients had a preoperative computed tomography (CT) scan, and a chest X-ray taken immediately after the operation. The CT studies were performed with a 16-slice scanner (Siemens SOMATOM Sensation 16) using the protocol described in our previous study (Chang et al., 2006, 2007). Informed consent was obtained from the guardian of each patient, and this study was approved by our hospital's ethics committee (CGMH: IRB 94-934B).

An appropriately simplified FEA model is necessary for a complicated analysis such as this to ensure that the model is easily reproducible while still generating useful ideas to check the possible implications and complications of the operation. Our FEA models consisted of the ribs, sternum, and costal cartilage, which provide the major contributions to rib cage integrity. Since the contributions to the chest wall integrity of the muscles and skin were much less than those of the bone and cartilage, we ignored those components in our study (Yang and Wang, 1998; Gignac et al., 2000; Feng et al., 2001). Moreover, based on our clinical observations, the shape of the vertebral column of each of the three patients did not significantly change after the Nuss procedure. Therefore, the displacement of the joints between the rib and spine was assumed to be constant, and the vertebral columns were not included in our rib cage models.

Table 1
Biomechanical data from the three simulations of the Nuss procedure

	Case 1	Case 2	Case 3
Sex/age	M[b]/8	F[b]/7	M[b]/7
Pectus index	5.3	4.7	5.2
Elevation of the end of sternum (cm)[a]	4.47	3.40	3.90
Simulation results:			
1. Simulation displacement at the end of the sternum (cm)	4.49	3.43	3.90
2. Loading force at the end of the sternum (N)	140	120	190
3. Maximal stress at the end of the third rib near the spine (MPa)	24.89	48.50	33.25
4. Elongation of the right fourth cartilage (%)	4.30	5.74	5.26

[a] The elevation at the end of the sternum was measured as the difference in distance from the inner margin of the sternal end to the anterior surface of the vertebral body based on the preoperative CT scan and postoperative X-ray film.
[b] M, male; F, female.

Since the grayscale value of costal cartilage was indistinct from other tissues on the CT slices, the automatic segmentation of cartilage by the AMIRA visualization software was incomplete. To overcome this problem, we developed a semiautomatic procedure for reconstructing the rib cage model. First, the CT slices were imported into AMIRA, and the segments of the rib, sternum, and costal cartilage were labeled automatically by assigning respective grayscale values to them. Then, the segments of costal cartilage were modified manually. These segment modifications relied on our professional experience.

After the geometric models were created, the FEA models were generated with tetrahedral elements using AMIRA. A convergence test was performed to confirm the simulated accuracy of the mesh by comparing the simulated results of six rib cage meshes (Table 2). The convergence criterion was that the difference in the corrected displacement at the end of the sternum was less than 1%, and the final choice was a FEA model that consisted of approximately 320,000 tetrahedral elements.

2.2. Finite element analysis

The material properties of bone and cartilage were based on previous articles (Granik and Stein, 1973; Yang and Wang, 1998), and an elastic modulus of 11.5×10^9 Pa was chosen for the ribs and sternum. We assumed a value of 12.25×10^6 Pa for the pectus costal cartilages, which was about half the value of normal cartilage, as suggested by Feng et al. (2001). The material properties of the ribs, sternum, and costal cartilage were modeled as linear, as described by Murakami et al. (2006) and Roberts et al. (2005). During the Nuss procedure, the concave side of the Nuss bar was placed under the sternum and then forcibly turned and rotated to raise the depression of the sternum. Although this turning process may cause some damage to the chest wall, we ignored it in our analysis.

Table 2
Convergence test of six rib cage meshes

Number of elements	97,939	170,660	223,522	269,163	320,519	443,470
Corrected displacement at the end of the sternum	4.304	4.370	4.417	4.462	4.490	4.519
Mean difference (%)		1.5381	1.064	1.035	0.619	0.649