

# The effect of model errors caused by discretisation of the identification process of inhomogeneous materials

**Citation for published version (APA):**

Kleintjens, L. J. S. (1993). *The effect of model errors caused by discretisation of the identification process of inhomogeneous materials*. (DCT rapporten; Vol. 1993.098). Technische Universiteit Eindhoven.

**Document status and date:**

Published: 01/01/1993

**Document Version:**

Publisher's PDF, also known as Version of Record (includes final page, issue and volume numbers)

**Please check the document version of this publication:**

- A submitted manuscript is the version of the article upon submission and before peer-review. There can be important differences between the submitted version and the official published version of record. People interested in the research are advised to contact the author for the final version of the publication, or visit the DOI to the publisher's website.
- The final author version and the galley proof are versions of the publication after peer review.
- The final published version features the final layout of the paper including the volume, issue and page numbers.

[Link to publication](#)

**General rights**

Copyright and moral rights for the publications made accessible in the public portal are retained by the authors and/or other copyright owners and it is a condition of accessing publications that users recognise and abide by the legal requirements associated with these rights.

- Users may download and print one copy of any publication from the public portal for the purpose of private study or research.
- You may not further distribute the material or use it for any profit-making activity or commercial gain
- You may freely distribute the URL identifying the publication in the public portal.

If the publication is distributed under the terms of Article 25fa of the Dutch Copyright Act, indicated by the "Taverne" license above, please follow below link for the End User Agreement:

[www.tue.nl/taverne](http://www.tue.nl/taverne)

**Take down policy**

If you believe that this document breaches copyright please contact us at:

[openaccess@tue.nl](mailto:openaccess@tue.nl)

providing details and we will investigate your claim.

The Effect of Model Errors Caused by Discretisation on  
the Identification Process of Inhomogeneous Materials.

rapportnr. WFW 93.098

Lucien Kleintjens  
Juli 1993

Stageverslag

Begeleiders: ir. M.R. van Ratingen  
dr. ir. C.W.J. Oomens

Technische Universiteit Eindhoven  
Vakgroep Fundamentele Werktuigkunde

---

# **The Effect of Model Errors Caused by Discretisation on the Identification Process of Inhomogeneous Materials.**

## **Contents**

### **Summary**

- 1. Introduction**
  - 2. Theory of the Identification Method**
  - 3. Setup of the Simulations**
  - 4. Estimation Results**
    - 4.1 Data and Models Used for the Estimation Process**
    - 4.2 Results with a 10x10 Mesh**
    - 4.3 Results with a 20x20 Mesh**
  - 5. Conclusions**
  - 6. Recommendations**
- References**
- Appendix The Neu2dat Program**

## Summary

The identification method is a mixed numerical-experimental method used to determine the properties of an inhomogeneous material. For this method a finite element model of the experimental configuration is needed. The aim of this study is to investigate the influence of discretisation on the estimation process. Therefore an experiment with known parameter values is simulated and used to generate the experimental data needed for the parameter estimation process, which is performed using the same model with a less refined mesh. Although the estimated parameters generally converge, this does not guarantee an acceptable solution for all meshsizes.

### 1. Introduction

For inhomogeneous materials, like human skin, standard characterization methods provide limited possibilities because of the homogeneous strain field required. An experimental-numerical testing method does not have this limitation, since the actual strain field is taken into account. Hendriks (1991) proposed an experimental-numerical approach called the identification method. The objective of this method is to determine a number of unknown parameters from an appropriate constitutive model for some material. Its principle is shown in figure 1.

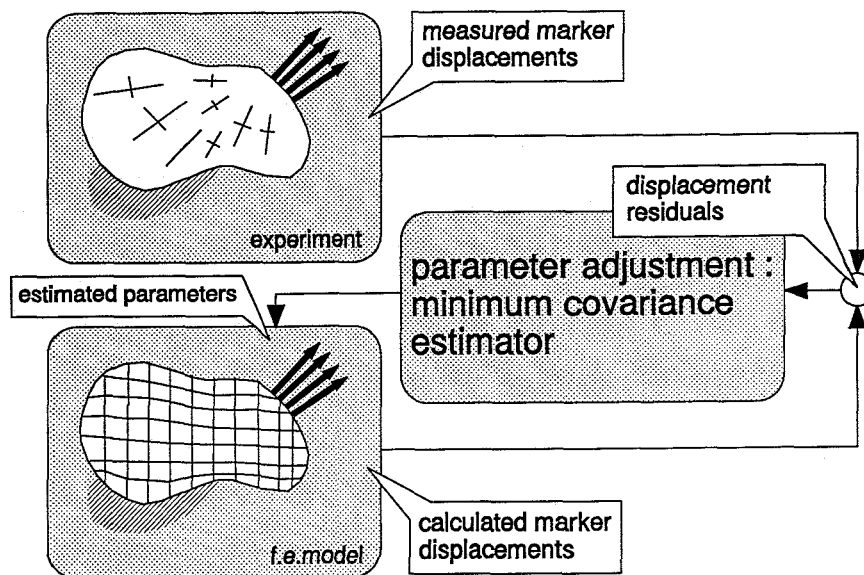


Figure 1: Diagram of the identification method

Some specimen is loaded in such a way that an inhomogeneous strain field exists in the specimen. The inhomogeneous strain field at the surface of the sample is measured using a digital

imaging technique. With a finite element model of the experimental configuration a strain field is computed, and compared with the actual strain field. The weighted difference is then used to adjust the estimated parameters. This process is repeated for several iterations until convergence is gained. For flexibility with respect to the used models a module PAREST (Courage, Hendriks, 1989) was written for the finite element code DIANA which allows an automatic call of DIANA to update parameters.

The identification process as described above is influenced by various errors in the model which is chosen to describe the experiment. Some of these model errors are caused by discretisation of the experimental domain. In this study the effect of these errors is investigated. This is done by simulating the experiment of which the parameters have to be estimated with a relatively fine mesh. The data acquired from this simulation is then, possibly after adding a measurement error, used for the estimation process, but now using a coarser mesh.

In Section 2 the identification method is briefly described. Section 3 describes the setup of the simulations and the estimations. In Section 4 the results for the different tests are given and discussed. Conclusions and recommendations are given in Section 5 and 6 respectively.

## 2. Theory of the Identification Method

This section gives a brief description of the identification method. For a more detailed study we refer to Hendriks (1991). The algorithm is based on a sequential minimum variance approach. The non-linear mathematical problem can be described as follows:

$$\mathbf{y} = \mathbf{h}(\mathbf{x}) + \mathbf{v} \quad (1)$$

With  $\mathbf{y}$  being a column containing the observed displacements,  $\mathbf{x}$  the set of material parameters to be estimated and  $\mathbf{v}$  a column of observation errors.  $\mathbf{h}(\cdot)$  describes the dependence of the observations  $\mathbf{y}$  on  $\mathbf{x}$ . The iterative scheme, used to estimate  $\mathbf{x}$  is given by

$$\mathbf{x}_{i+1} = \mathbf{x}_i + \mathbf{K}_{i+1} (\mathbf{y} - \mathbf{h}(\mathbf{x}_i)) \quad (2)$$

with  $\mathbf{K}_{i+1} = (\mathbf{P}_i + \mathbf{Q}) \mathbf{H}_{i+1}^T (\mathbf{R} + \mathbf{H}_{i+1} (\mathbf{P}_i + \mathbf{Q}) \mathbf{H}_{i+1}^T)^{-1} \quad (3)$

$$\mathbf{P}_{i+1} = (\mathbf{I} - \mathbf{K}_{i+1} \mathbf{H}_{i+1}) (\mathbf{P}_i + \mathbf{Q}) (\mathbf{I} - \mathbf{K}_{i+1} \mathbf{H}_{i+1})^T + \mathbf{K}_{i+1} \mathbf{R} \mathbf{K}_{i+1}^T \quad (4)$$

where  $i$  denotes the iterative step and

$$\mathbf{H}_{i+1} = \left( \frac{\delta \mathbf{h}(\mathbf{x})}{\delta \mathbf{x}} \right) \Big|_{\mathbf{x}=\mathbf{x}_i} \quad (5)$$

expresses the sensitivity of the model output for parameter variations. To restrict the model's influence when parameter errors are large, the sum of squares is weighted with matrices  $\mathbf{P}$  and  $\mathbf{Q}$ .  $\mathbf{P}_i$  is updated in a way that leads to an optimal minimum variance estimator in the linear case. In practice  $\mathbf{Q}$  prevents the parameter error covariance  $\mathbf{P}_i$  to become too small.

The matrix  $\mathbf{R}$  indicates the confidence in each separate measurement, so that more accurate

measurements dominate the gain matrix  $\mathbf{K}_i$ .

The estimator as described above is implemented in the extra DIANA module PAREST and has been used for the tests in Section 4.

### 3. Setup of the Simulations

The experiments used for the estimation of the different geometric and material parameters are numerically simulated. The sample used for the simulations is a flat membrane (100 x 100 x 2), the material of which is orthotropic and linear elastic with material parameters  $\mathbf{E}_1$ ,  $\mathbf{E}_2$ ,  $\mathbf{G}_{12}$  and  $\mathbf{n}_{12}$  and the material 1-axis depending on the global coordinates. The fiber layout is described by a bilinear function

$$\alpha = \mathbf{b}_1 + \mathbf{b}_2\mathbf{x} + \mathbf{b}_3\mathbf{y} \quad (6)$$

where  $\alpha$  denotes the angle between the material 1-axis and the global x-axis. A 40 x 40 mesh of four-noded plane stress elements is used for the experiments. The geometric and material parameters used are as follows:

$\mathbf{b}_1 = 0.2$	$\mathbf{E}_1 = 0.3$
$\mathbf{b}_2 = 0.004$	$\mathbf{E}_2 = 0.1$
$\mathbf{b}_3 = 0.002$	$\mathbf{G}_{12} = 0.1$
	$\mathbf{n}_{12} = 0.3$

In figure 2 the fiber layout for these parameters is shown.

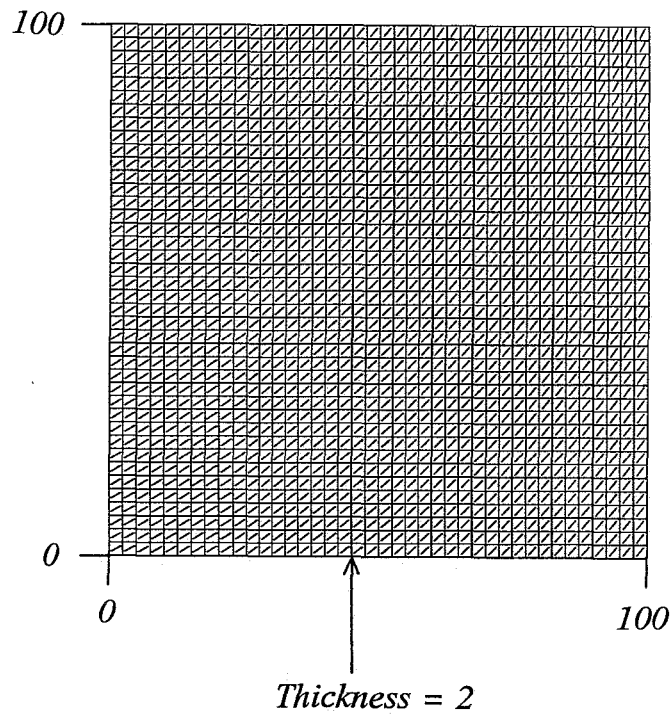


Figure 2: Direction of the material 1-axis in the sample

The sample is loaded with three different sets of boundary conditions. How the sample was loaded in each of the three cases is shown in figure 3.

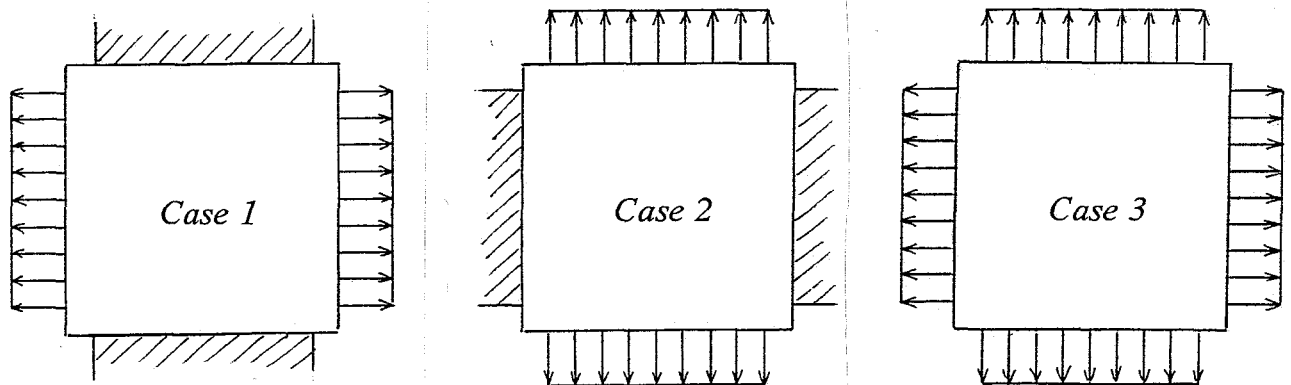


Figure 3: Prescribed displacements for all loadcases

## 4. Estimation Results

For the parameter estimation process, the same material and the same type of function to describe the fiber direction is assumed as for the experiments. This means that the only model error introduced in the parameter estimation model is caused by differences in meshsize.

### 4.1 Data and Models Used for the Estimation Process

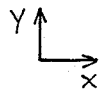
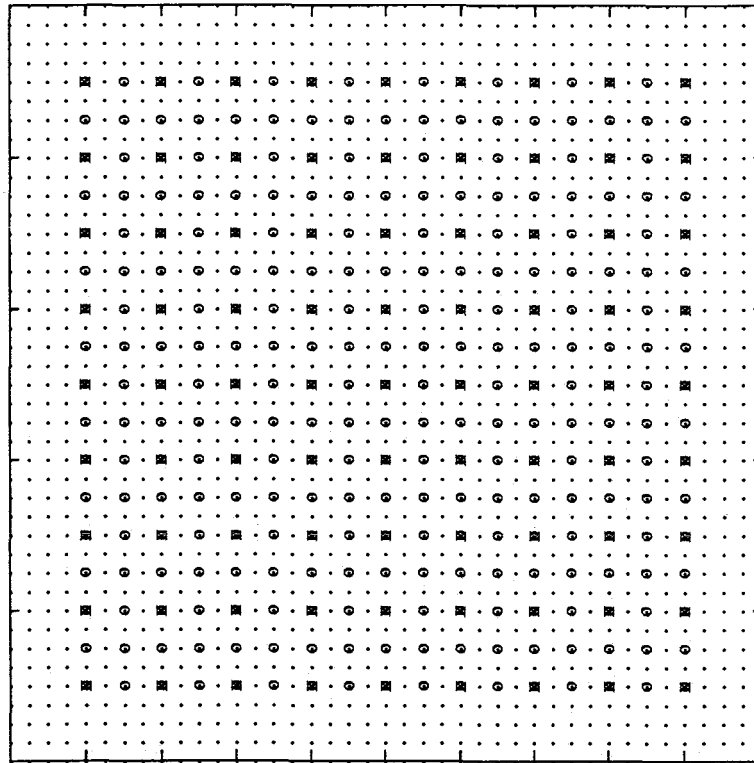
The nodal displacements for each of the loadcases are acquired from the simulated experiment as described in Chapter 3. However, since different meshes are used for the parameter estimation, a data reduction procedure was performed. From the original 1681 nodal displacements of the 40 x 40 mesh used for the experiment, 289 and 81 were used for the estimations which were performed with 20 x 20 and 10 x 10 element meshes respectively, as shown in figure 4. The material 1-axis with the exact parameters for the reduced meshes are shown in figure 5.

Again the material is assumed to be orthotropic and linear elastic with material parameters  $E_1$ ,  $E_2$ ,  $G_{12}$  and  $n_{12}$ . The material 1-axis is described with a bilinear function

$$\alpha = b_1 + b_2x + b_3y \quad (7)$$

It is not possible to estimate both  $E_1$  and  $E_2$ , since only displacements are given as boundary conditions, which leaves only the possibility to estimate the stiffness ratio  $E_1/E_2$ . In these tests  $E_1$  is set to the exact value used for the numerically simulated experiments as described in Section 3. For the tests with the 10 x 10 mesh,  $G_{12}$  was also set to the exact value.





■ = Observations for 10x10 Mesh Estimation

■ ◦ = Observations for 20x20 Mesh Estimation

Figure 4: Nodal displacements used for parameter estimation

The following values are given as initial guesses for the parameters which are to be estimated:

$$\begin{array}{ll}
 \mathbf{b}_1 = 0.0 & \mathbf{E}_2 = 0.2 \\
 \mathbf{b}_2 = 0.002 & \mathbf{G}_{12} = 0.2 \\
 \mathbf{b}_3 = 0.004 & \mathbf{n}_{12} = 0.2
 \end{array}$$

Different loadcases are being applied during the iteration proces, since this may give an indication on the severeness of the model error (van der Burgt, 1992), in this case caused by the

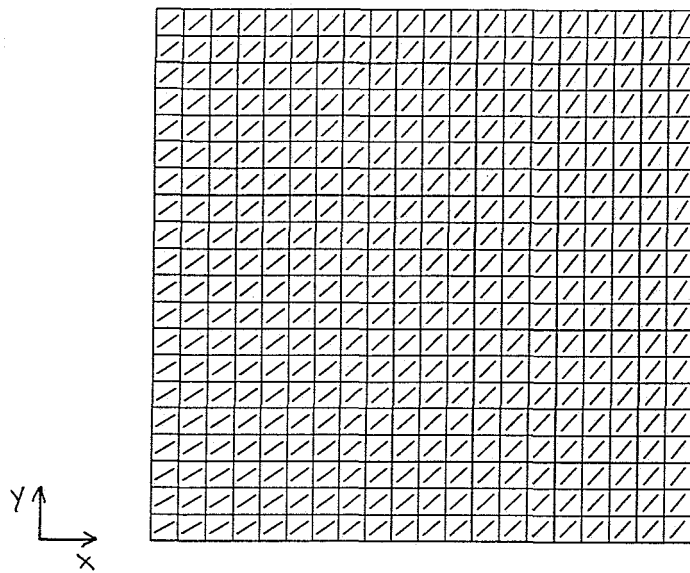
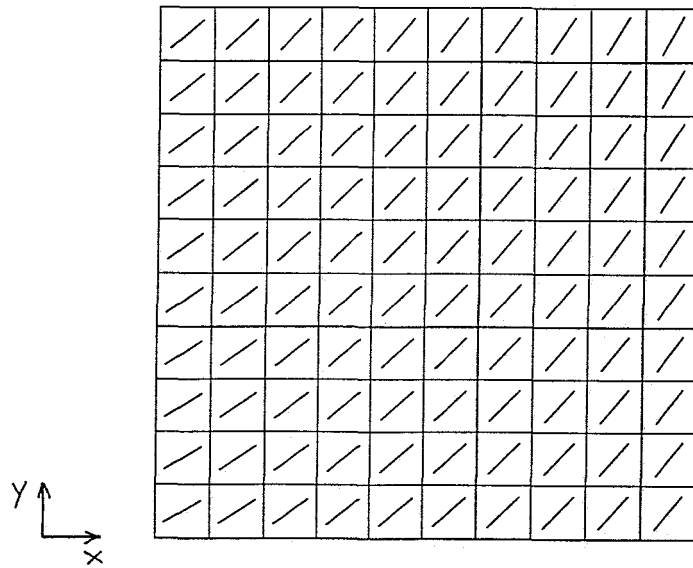


Figure 5: Material 1-axis for reduced meshes

discretisation. 15 iterations are executed, after 5 and 10 iterations a different observation column  $y$  and  $h(x)$  is used in the estimation process.

## 4.2 Results with a 10x10 Mesh

The following results are achieved using a 10 x 10 element mesh to compute the displacement field. 81 nodal displacements are taken into account in the observation vector  $y$ .

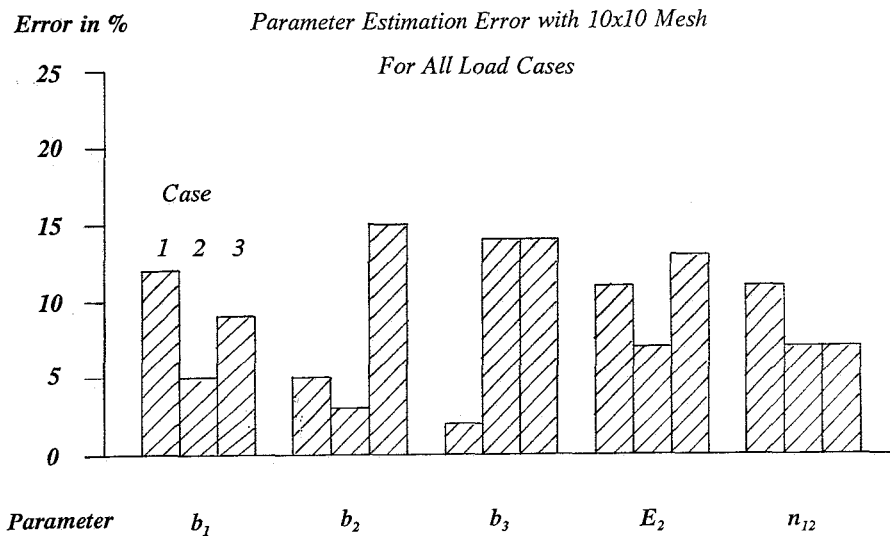
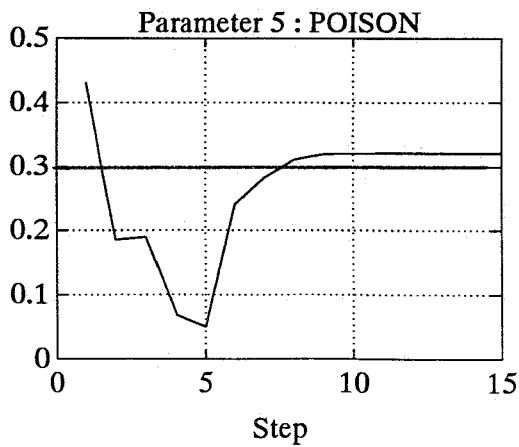
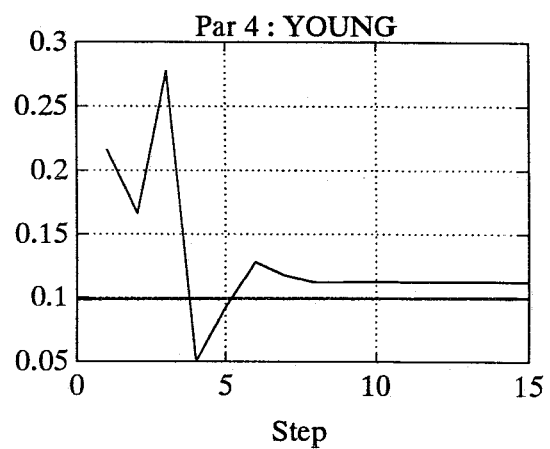
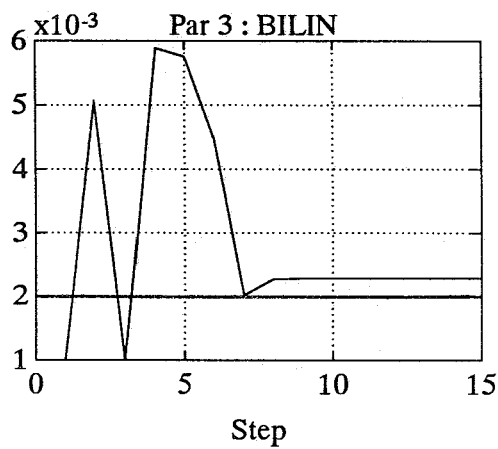
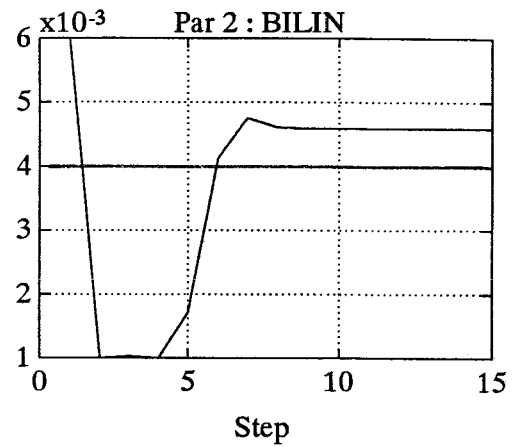
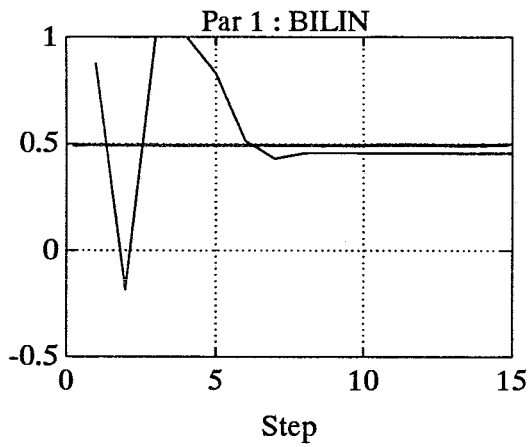


Figure 6: Estimation error with 10x10 mesh

The results for case (1 1 1) and (2 3 1) are also shown in figure 7. It is clear that the parameters all converge to a stable value within 15 iterations. However, the value to which the parameters converge is depending on the (final) loadcase which is being applied. This is caused by the presence of model errors.

The residual displacement field for case (1 1 1) is shown in figure 8. There is a clear structure visible in the residuals, with the biggest deviations from the experimental displacement field close to the corner elements.

This is easily understood if we look at the difference between the exact solution of the displacement field for the 40 x 40 mesh, compared to that of the 10 x 10 mesh computed with the same parameters, shown in figure 9. It is clear that the difference between these two solutions, solely caused by the different meshes, is mainly present in the same regions close to the corner elements. The presence of the bigger deviations from the exact solution near the corners can be explained by the high strain gradients which appear in these regions, as can be



$$\begin{aligned}
 b_1 &= 4.555 \cdot 10^{-1} \\
 b_2 &= 4.592 \cdot 10^{-3} \\
 b_3 &= 2.287 \cdot 10^{-3} \\
 E_2 &= 1.126 \cdot 10^{-1} \\
 n_{12} &= 3.213 \cdot 10^{-1}
 \end{aligned}$$

Figure 7: Parameter estimations for each iteration

### Displacement Residual Field (x41)

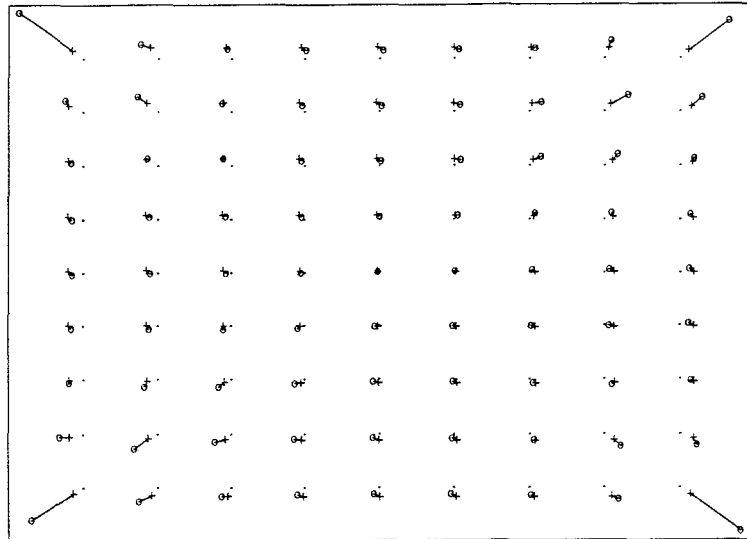


Figure 8: Residual displacements for 10x10 mesh

seen in figure 10.

The next tests are performed with the nodal displacements of the corner elements discarded. In the column of observations  $y$  77 nodal displacements remain. The results are compared to those of the previous tests and can be found in figure 11.

It is clear that the estimation of the geometric parameters  $\mathbf{b}_n$  and of  $\mathbf{E}_2$  improves, but  $n$  is very badly estimated in these cases. The residual displacements for case (1 2 3) can be found in figure 12. The residuals have become smaller, still with the biggest discrepancy near the corner elements. The bad estimation of  $\mathbf{n}_{12}$  when the nodal displacements at the corners are not used as observations is caused by the absence of large inhomogeneities in the remaining displacement field. In fact this displacement field approaches that of a perfect biaxial test and it is evident that a local approach with mere kinematic boundary conditions does not allow the estimation of  $\mathbf{n}_{12}$  in such a test. The homogeneity of the strain field can also be seen in the strain field in figure 10.

### 4.3 Results with a 20x20 Mesh

Some more tests were performed using a 20 x 20 mesh to estimate the displacement field. 289 nodal displacements are taken into account in the observation vector  $y$ .  $\mathbf{G}_{12}$  is also estimated. The results are computed for case (1 2 3) and compared to those of the 10 x 10 mesh for the same case. In a second test an observation error was introduced (white noise,  $\sigma = 10^{-3}$ ) on the observed displacements. The results can be found in figure 13.

### *Displacement Residual Field (x35)*

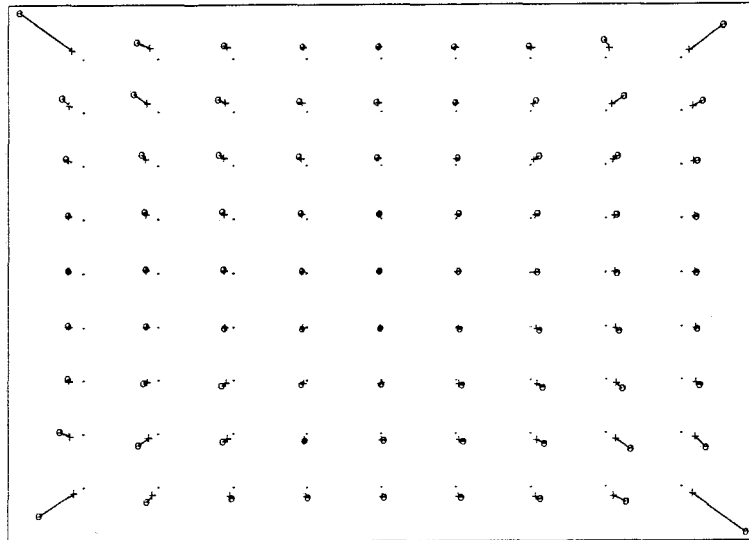
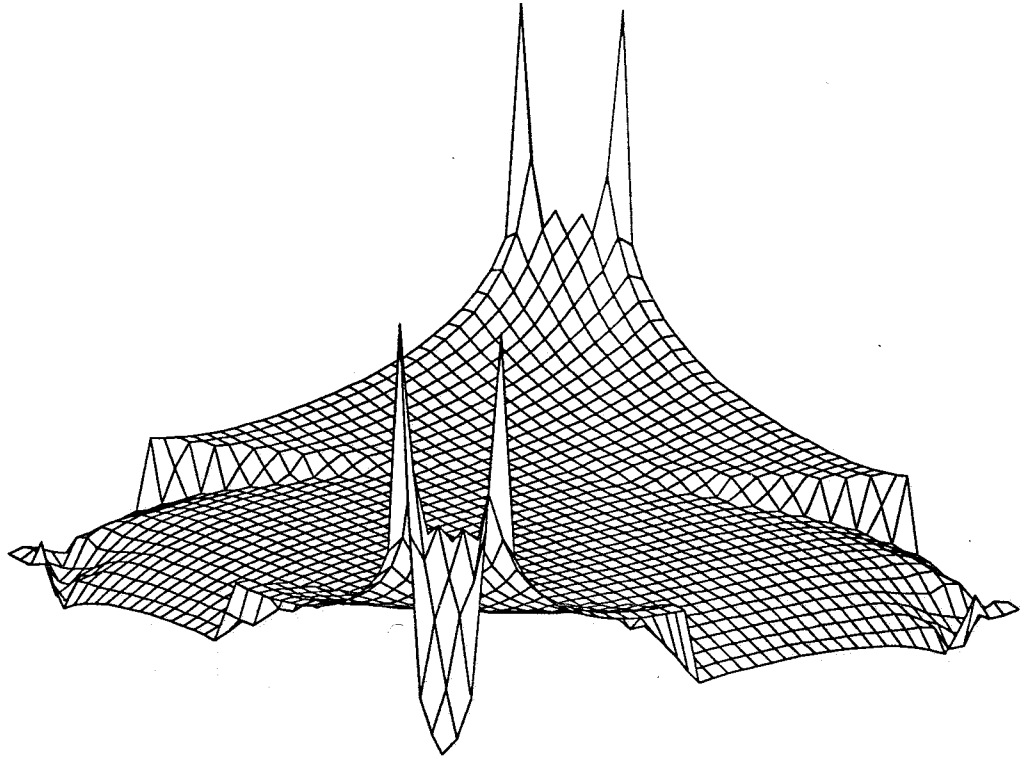


Figure 9: Difference in computed displacements between 10x10 and 40x40 meshes

It is clear that the use of a finer mesh for the parameter estimation gives better results. The estimation of the geometric parameters for the 20 x 20 mesh is better than that of the 10 x 10 mesh without corner nodes. The estimation of  $\mathbf{E}_2$  is in the same order of magnitude. The estimation of  $\mathbf{n}_{12}$  is also better, but does not improve compared to the original 10 x 10 mesh estimation. This probably means that the experimental setup does not give a sufficiently inhomogeneous strain field to provide enough information for a good estimation of  $\mathbf{n}_{21}$ . The residual field for the 20x20 mesh for this case can be found in figure 14.

Introducing an observation error on the nodal displacements gives a slightly worse estimation for all parameters. Again,  $\mathbf{n}_{12}$  is effected most, as can be seen in figure 15. This is another indication that little information for estimating  $\mathbf{n}_{12}$  is present in the observed displacement field and therefore easily disturbed.

Principle Strain E1 for Loading Case 3



Principle Strain E2 for Loading Case 3

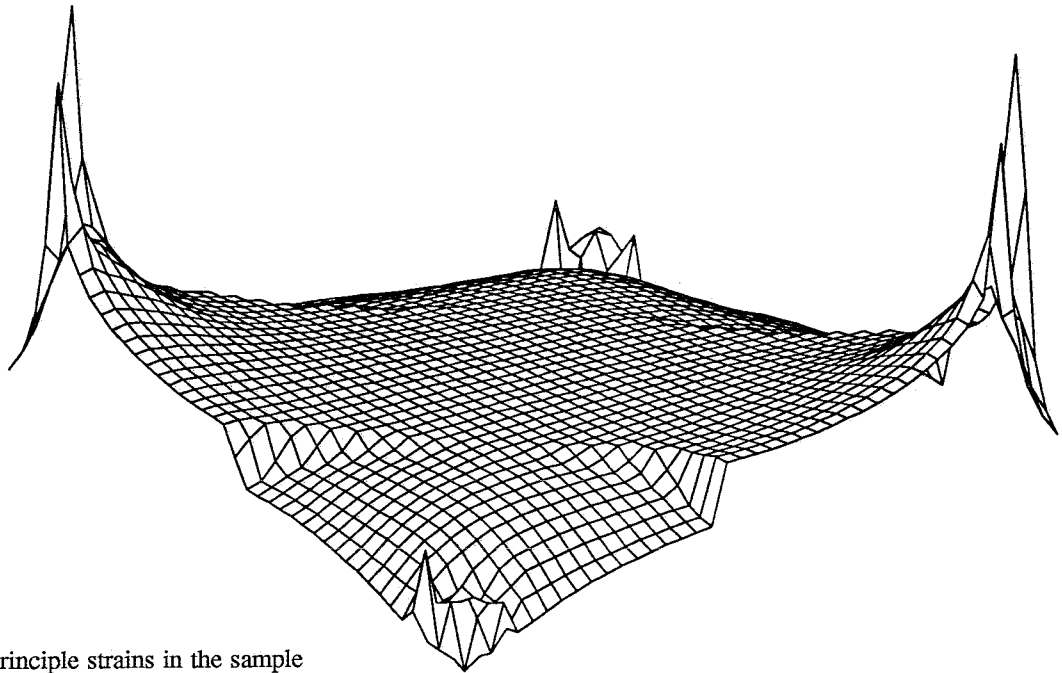


Figure 10: Principle strains in the sample

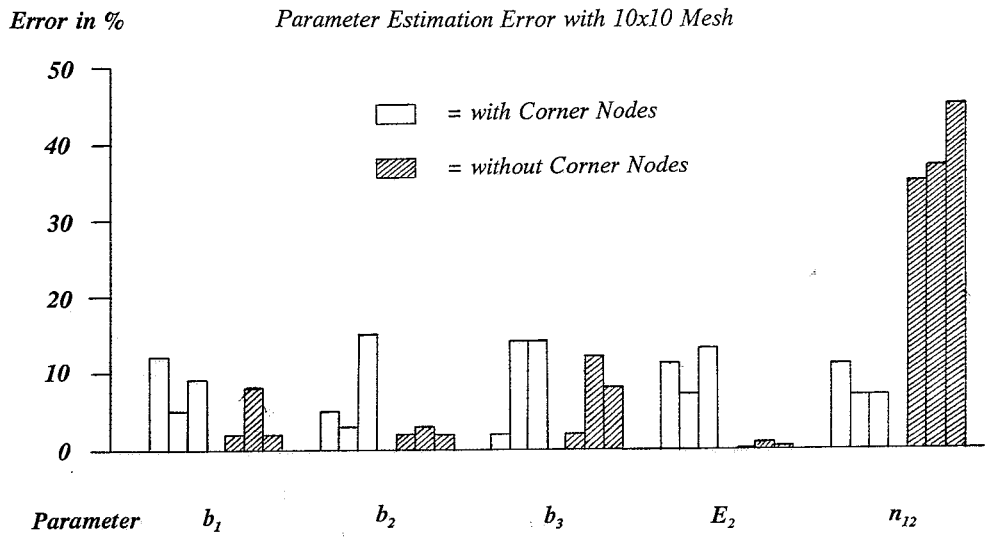


Figure 11: Estimation error with 10x10 mesh without corner nodes

*Displacement Residual Field (x174)*

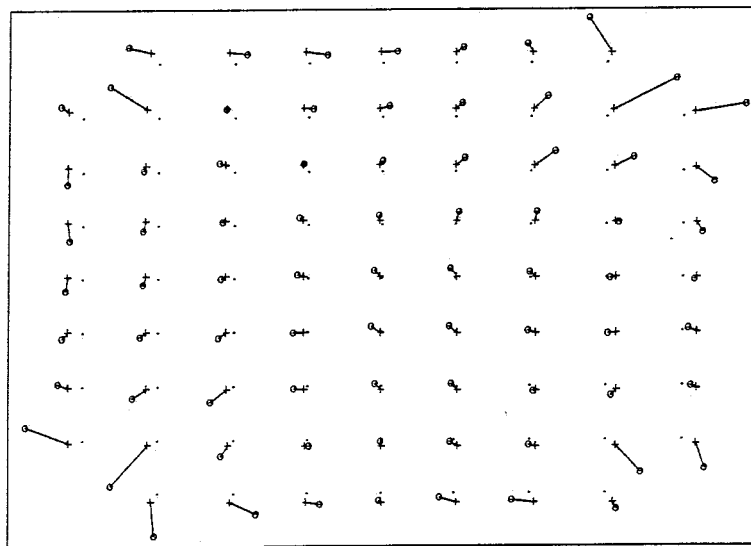


Figure 12: Residual displacements for 10x10 mesh without corner nodes



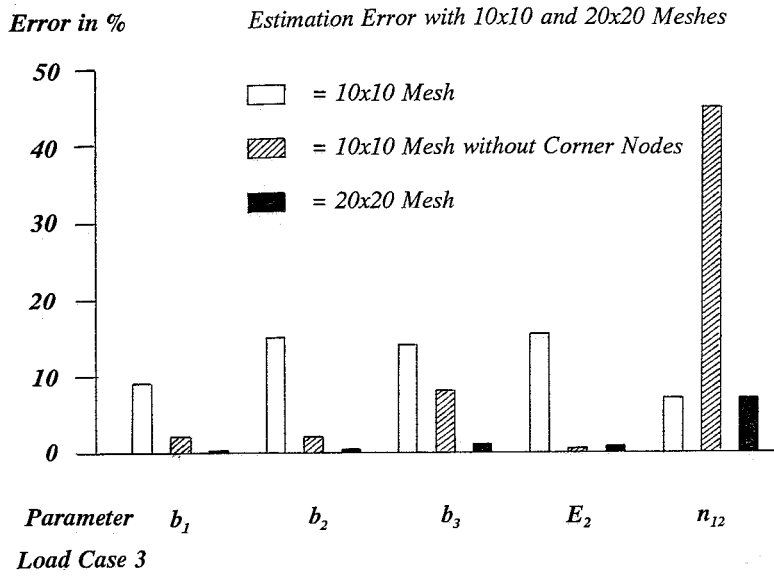


Figure 13: Estimation error with 20x20 mesh

**Displacement Residual Field (x338)**

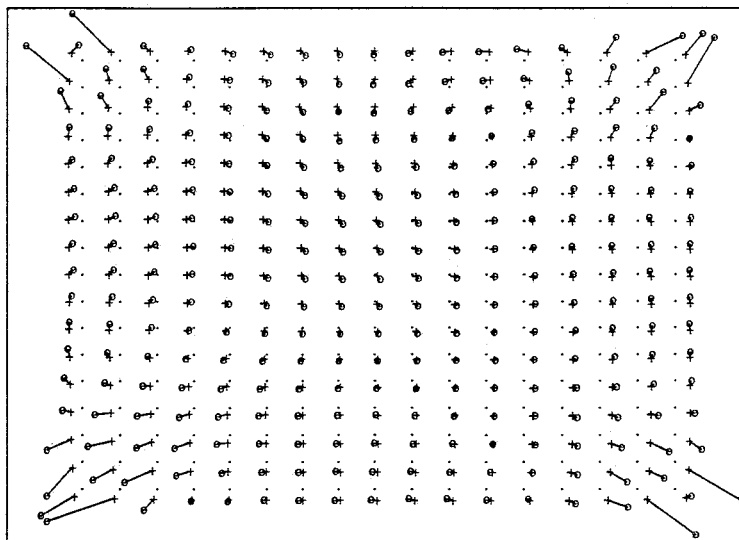


Figure 14: Residual displacements for 20x20 mesh

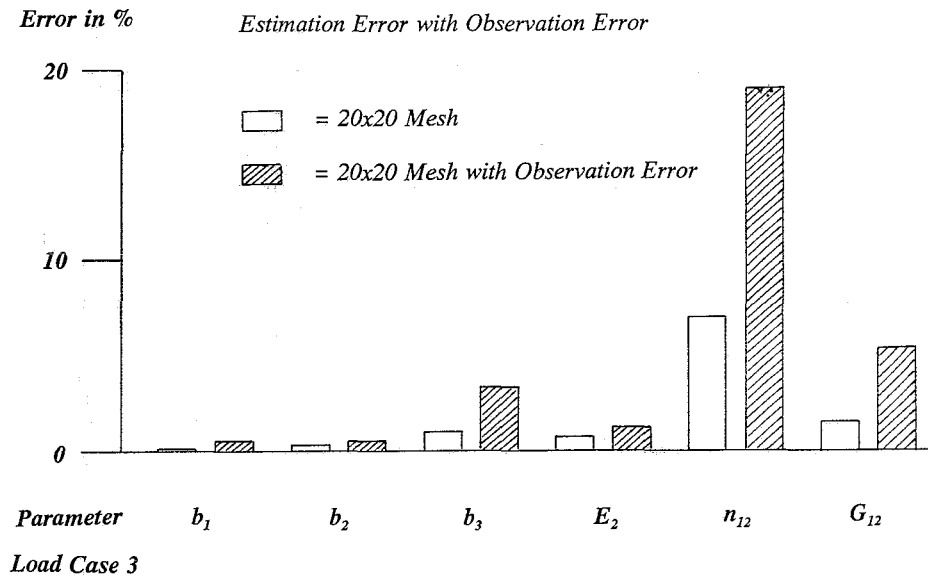


Figure 15: Estimation error with 20x20 mesh and observation error

## 5. Conclusions

From the tests that were performed, we may conclude that it is not always possible to get good estimation results, even if the theoretical model assumed is correct. Discretisation errors can cause discrepancies between the parameters' real and estimated values. The use of a more refined mesh reduces this problem and therefore gives better results, but even for relatively fine meshes they may still be unsatisfactory. The estimation of  $n_{12}$  may still cause problems if the testing conditions do not yield a sufficiently inhomogeneous strain field, or the inhomogeneity is primarily present in regions with large strain gradients.

## 6. Recommendations

It is recommended to look closely to the error introduced by the discretisation process. The use of locally refined meshes in regions with large strain gradients may reduce the problem of estimating  $n_{12}$  and give better results in general. It is a good idea to compare the solutions for the same parameters of finite element models with different meshsizes for the loadcases to be investigated, to see which regions of the displacement field may cause problems.

## References

Burgt, T.C.G. van der, Effect of Different Loadcases in the Identification Process of Inhomogeneous Materials. Internal report WFW 92.115, Eindhoven University of Technology, 1992.

Courage, W.M.G., Hendriks, M.A.N., Module Parest. Internal Reports WFW 89.38 and WFW 89.39, Eindhoven University of Technology, 1989.

DIANA Finite Element Analysis, Release 5.0. TNO Building and Construction Research, 1992.

Hendriks, M.A.N., Identification of the Mechanical Behaviour of Solid Materials. Ph.D.-thesis, Eindhoven University of Technology, 1991.

Meuwissen, M.H.H., A Non-Invasive Method to Find Langer's Lines of Skin. Internal report WFW 92.055, Eindhoven University of Technology, 1992.

Ratingen, M.R. van, Identification of Orthotropic Membranes with High Stiffness Ratio. Eindhoven University of Technology, 1992.

## Appendix The Neu2dat Program

This section will give a brief description of the program 'neu2dat', which was written to extract data from so-called 'neutral files', generated by the DIANA finite element package, and convert these data into tables which can be put in the DIANA inputfile.

### Starting the Program

The program can be started with the command 'neu2dat'. The most comfortable way to use the program is to copy both your neutral file and the 'neu2dat' command file into the same directory. After starting the program, three filenames are requested:

-- The neutral filename

This is the name of the neutral file you want to extract data from.

-- The output filename

This is the name of the ASCII file to which the extracted data will be written. The program will warn you if a file with this name already exists. If you do not change the name of the output file, the data will be appended to the existing file.

-- The input filename

This file is used by the program for certain data selection or processing routines. At this moment the file is only used for the selection of records out of your neutral file. In this case the file should contain a list of the records you want to select out of a certain dataset. The file can be typed in by hand or generated by e.g. a fortran program (see the file 'node8.f').

### Selection of the Data

After entering the filenames the neutral file will be opened and a list of the available datasets will appear. A description of the possibly available datasets is given in the DIANA manual. Choose one of the dataset numbers. You can now generate a table from the items (or compounds) which are available in this dataset.

A list of all the items available in the selected dataset will appear. Again, choose one (or more)

item(s) which you want to have listed in the output file. Enter 0 if you don't want to select any more items.

Some items contain different 'compounds'. These can generally be recognized if the itemname end with an 'i'. If the item you choose is a compound item, another list of the available compounds will appear, which can be selected in the same manner as the items.

## Selection of the Processes

After selecting the items and compounds to be listed, it is possible to select some data processing routines, which change or expand the data which will be written to the output file.

First, a list of so-called 'general processes' appears. These processes are not performed on one specific item or compound, but on the entire table. At this moment the following options are available:

1 -- Write table heading

At the top of the table some information will be given concerning the data which is listed in the table. The header is built up as follows:

```
[dataset]
[#] [itemlcmp] [itemlcmp] ...
```

If compounds have been selected, the name of the compound is given in the table header. A #-sign in the first column of the header indicates that line numbers have been added to the table (See general process 3).

2 -- Records from input file

Only the records which are listed in the input file are written in the table in the output file. (Very comfortable for data reduction, e.g. if you do not want the nodla displacements in all nodes to be listed). The input file should have the following structure:

```
[rec #]
[rec #]
.
.
.
[rec #]
```

The maximum size of the input field is 5 (i.e. record numbers up to 99999). Record numbers

should be integers, numbers higher than available in the neutral file will probably generate an error message.

### 3 -- Number lines in output file

Every line in the output file will be preceded by a line number. This is not the same as the record number, at least if general process 2 or process 2 is in use !

One or more of the general processes listed above can be selected. You can leave the menu by entering a 0 value.

After selecting the general processes, you will be asked to select one or more processes for each selected item. At this moment two options have been implemented:

### 1 -- Add measurement error to item

Add a measurement error will be added to the values of the items. The standard deviation of the measurement error has to be entered. The measurement error is a randomly generated white noise (normal distribution, average = 0). The item should be of the double precision type.

### 2 -- Filter on item

This option offers the possibility to select certain data out of the neutral file, based on a selection criterion for one or more of the selected items. First the datatype of the item is requested. This can be (i)nteger, (r)eal or (s)tring. After this the minimum and maximum values of the items which should be selected are requested.

Warning ! The filter will always affect the whole table, although filter conditions might be only be selected for one item of the table. If more than one filter is selected, only those records will be listed which meet all filter conditions.

Again, you can leave the 'process' menu by selecting 0.

## The End of the Program

The data of the selected items will now be read and, after executing the selected processes, written to the output file in ASCII form.

After this, the program will ask you if you want to add some more tables to the same file. You

can choose a different dataset or a different neutral file to read from, and also a new input file to generate the table. If you don't want to continue the program will be terminated.

## Info about Neu2dat

Filename:	neu2dat*	(executable)
Home Directory:	/users/sg4/parest/lucien/utilities	
Other Files:	neu2dat.f	(source code)
	neu2dat.man	(Dutch manual)
	node8.f	(example of input file generation program)
	node8.inp	(example of input file)