A Systematic Methodology for the Generation of SPICE Models Feasible for EMC Analysis
Abstract

This document describes different simulation and measurement methods for the characterisation of barriers for the purpose of EMC analysis. The different methods are compared to each other in order to establish guidelines for when a particular method is preferable. The goal has been to describe the studied barriers with a circuit diagram that can be analysed in a standard circuit analyser program such as SPICE. Thus, engineering models that can be used by a large group of users have been produced.

In order to produce the circuit diagrams, both commercially available 3D programs and a developed 2D program have been used. Circuit diagrams produced by these programs have then been used in SPICE. The results have been compared with measurements done with a vector network analyser in order to verify the methods and models.

Barriers that cannot easily be characterised by numerical methods, such as filters with unknown components, a different approach had to be used. For these types of barriers a measurement was first performed and then a developed program for adapting the calculated scattering parameters for a circuit to the measured was used. In this way a circuit diagram for the measured barrier was determined.

Key words: Conducted emission, Conducted immunity, Crosstalk, EMC analysis, PCB, Printed circuit board, SPICE simulation

SP Sveriges Provnings- och Forskningsinstitut
SP Rapport 2000:07
ISBN 91-7848-804-4
ISSN 0284-5172
Borås 1999

SP Swedish National Testing and Research Institute
SP Report 2000:07

Postal address:
Box 857, SE-501 15 BORÅS, Sweden
Telephone +46 33 16 50 00
Telex 36252 Testing S
Telefax +46 33 13 55 02
Contents

Abstract 2

Contents 3

Preface 5

Summary 6

1 Introduction 7

2 Barrier concept 9
  2.1 Types of barrier 9
  2.2 Studied barriers 10

3 Methodology used in the project 11

4 Characterisation of barriers 13
  4.1 Characterisation of barriers through measurements 13
  4.1.1 S-parameters for a two-port 14
  4.1.2 Transfer impedance 14
  4.1.3 Fixture for measuring transfer impedance for connectors 15
  4.2 Characterisation of barriers through simulations 16
  4.2.1 S-parameters for a multiport 16
  4.2.2 3D-simulations 17
  4.2.3 2D-simulations 18
  4.3 Generation of SPICE circuit based on measured S-parameters 24
  4.4 Transforming S-parameters to total voltages 26
  4.4.1 Simplified transformation for two-port measurements 28

5 Creation of SPICE circuit files 29
  5.1 Creation of netlist 29
  5.2 Creation of models 29
  5.3 SPICE simulations 31

6 Verification of models 33
  6.1 Test PCB for evaluating crosstalk between traces 33
  6.2 Test PCB for evaluating filter components 34
  6.3 Test PCB with combined barriers 35

7 Results 37
  7.1 D-sub connector 37
  7.1.1 Measurements 37
  7.1.2 Simulations 38
  7.1.3 Comparison measured and simulated results 39
  7.2 Shielded D-sub connector 41
  7.2.1 Measurements and results 41
  7.3 Shielded cable 41
  7.3.1 Measurements 41
  7.4 Filters 42
  7.4.1 Measurements 42
  7.4.2 Simulations 43
  7.4.2.1 Filter L1 44
Preface

This project has lasted for two years and has been financially supported by NUTEK within the program “Nya Miljöanpassade Byggsätt för Elektronikproduktion”. ABB has contributed by designing and manufacturing printed circuit boards used in the project. (One of the partners, IMC, has during the course of the project changed name to ACREO).
Summary

Different methods to derive circuit representations for barriers such as filters, cables, connectors etc. are presented. The circuit consists of linear passive components and can therefore easily be used in a circuit analysis program, such as e.g. SPICE, for computing the wanted response either in the time or in the frequency domain.

In order to derive the circuit representation for barriers with known geometry and material properties a finite difference program, FD2D, has been developed. In using the program the cross-section of the device has to be defined, this can easily be done by using the CAD-like user interface. When the geometry and all material properties are defined the program can either generate a circuit representation or the S-parameters for a given frequency range. The validity of the generated circuit representation has been tested against measurements for several test cases presented in this document.

For other types of barriers with unknown geometry and/or material properties an alternative method has been developed. This method uses measured S-parameters and an initial guess of the circuit representation as the starting point. The next step is to adjust component values for the elements in the circuit in order to find the best possible match to the measured S-parameters. This method has been implemented in a computer program called S-Calc. The program has been tested on several surface mounted filters.
1 Introduction

Traditionally the work in the area of EMC has mostly been performed by means of laboratory measurements. Such measurements are often very time consuming and expensive, especially if prototypes have to be manufactured for testing in the early stages of a project. Special equipment and test facilities are also often needed and consequently the test has to be performed at special test houses and by qualified personnel. Due to these facts the EMC issues are often addressed very late in a project and restricted to only one qualifying measurement. Such a policy can turn out to be unwise since if modifications are needed the costs can be very high and also difficult to apply. In order to increase the chance of passing the EMC test and thereby avoiding expensive redesigns late in the project, simulations can be of great help. By performing simulations early in a project it is possible to choose the right design, in terms of EMC performance, from the beginning and avoiding problems later on in the project.

In order to simulate a complex system it has to be divided into smaller manageable parts. The coupling of electromagnetic energy from one part to another can be described by the characteristics of a barrier. Thus, by characterising the barrier we are able to study the behaviour and also be able to optimise its performance with respect to some criteria. If we know the characteristics for all barriers in a system we can put them together in a simulation and are thereby able to characterise the whole system. By doing this we can e.g. compute the propagation of a disturbance from one point in the system to any other point. In this project we have studied a number of barriers that are common in most systems and characterised them by different methods. The aim has been to develop adequate engineering design models for minimizing emission and maximizing immunity over barriers. Studied barriers have been described by a circuit diagram that can be imported to any standard circuit simulator. A selection of studied barriers have been characterised by simulations as well as measurements.

The computer codes that have been developed in the project, FD2D and S-Calc, are freeware and can be acquired from Jan Carlsson, SP.
2 Barrier concept

One convenient way of structuring the complex problem of analysing a system is to use electromagnetic topology [1]. In using this concept we have to divide the system into several smaller parts or zones depending on the electromagnetic environment in the zones. The boundary between these zones represents a barrier that have a certain attenuation of electromagnetic energy that possibly prevents electronic equipment in one zone from disturbing equipment in another zone. This barrier could e.g. be a physical shield such as a metallic cover in an electronic apparatus or it could just be the distance between different parts. One example of the latter would be the crosstalk between two traces on a printed circuit board. When considering conducted disturbances most often a filter represents the barrier.

The electrical performance of a barrier can be described in several different ways. The choice in this project has been to describe the barrier with a circuit diagram containing linear elements. One apparent advantage with this approach is that the performance of the barrier can be studied in an ordinary circuit simulator, such as e.g. SPICE [2], that most engineers have access to. The performance can then easily be determined either in the time or the frequency domain. If we have circuit diagrams for all barriers in the system under consideration we can put them together in the circuit simulator and perform simulations on the whole system. In this case we can e.g. compute the propagation of a disturbance from one point in the system to another and are thereby able to judge if we will get EMC problems or not.

One advantage of using the concept of barrier is that we can build-up a library of models for barriers that we often use in different projects. By this we can fast and easily analyse systems and judge if we have to change anything in order to reach EMC.

2.1 Types of barrier

Generally a barrier can be placed in either one of two groups:

- Physical barrier
- Geometrical barrier

With a physical barrier it is here meant a commercially available construction element that we don't have all information about, i.e. we don't have sufficient data for the component so that we can characterise it by simulations. A filter is an example of a component that very often has to be placed in this group. If we buy a filter we often don't know the values of the components inside the filter and if we do, we often don't know the exact placement of them. The only method to characterise such a component, with some confidence, is through measurements.

Geometrical barriers are barriers that we know the exact geometry and material properties for. One typical example of a barrier in this group would be traces on a printed circuit board. In this case we know the width of the traces, the exact location and all relevant data for the circuit board (material in the substrate, thickness etc.). This type of barriers can be characterised either through simulations or measurements.
2.2 Studied barriers

In order to study different types of barriers three printed circuit boards were manufactured. Two of the boards were designed for the study of individual barriers while the third was designed for the study of combined barriers. Details for the manufactured printed circuit boards can be found in section 6. Measurements and simulations have also been performed on other type of barriers than printed circuit boards. One example is a D-sub connector.

The studied barriers are:

- Crosstalk between traces on a printed circuit board, Figure 6:1
- Filters mounted on a printed circuit board, Figure 6:2
- Combined barriers on a printed circuit board, Figure 6:3
- Crosstalk between pins in a 9-pin D-sub connector
- Transfer impedance of a shielded D-sub connector
- Transfer impedance of a coaxial cable of type RG-58

All studied barriers have been characterised by circuit diagrams with linear elements. These circuits can be used in an ordinary circuit simulator such as SPICE in order to determine the responses either in the time or the frequency domain.
3 Methodology used in the project

Two classes of methods have been used in order to characterise the studied barriers, measurements and numerical simulations. Measurements have served as the reference that simulated results have been compared against. All measurements have been carried out with a vector network analyser, which gives amplitude as well as phase information. This is essential since both are required in order to characterise a complex network.

For physical barriers measurement is the only possible way of characterising the behaviour. For this group of barriers a special tool was developed in the project by which it is possible to deduce a network from measured data. The program, called S-Calc, takes measured scattering parameters and computes the same for an assumed network. Then the program seeks optimal values for components in the network so that a best fit is found. By this procedure the wanted network describing the measured device can be determined. Of course, the success of this method is only guaranteed if the network entered into the program actually can represent the device under consideration. Thus, the program requires that the user have basic knowledge of circuit theory and at least some experience. The basis of the S-Calc program is described in section 4.3.

The electrical characteristics for geometrical barriers are determined by the geometrical shape and material properties. For this group of barriers numerical simulations as well as measurements have been carried out. Two classes of simulation programs have been used, 3D and 2D programs. The 3D programs were all commercial programs while the used 2D program was developed in the project. One advantage of the 3D programs is that they, in principle, are able to model any structure. However, disadvantages of this type of programs are that the simulation time is very long and they are also very expensive, and thus perhaps not interesting for small and medium sized companies.

Many types of structures can be viewed as being two-dimensional or nearly two-dimensional and for these a simpler and faster 2D simulation program can be used. Main advantages by this type of programs are that they are fast, easy to use and not so expensive as the 3D programs. Details concerning 3D and 2D simulations can be found in sections 4.2.2 and 4.2.3, respectively.

All simulation programs used in the project have the capability of producing either scattering parameters or equivalent circuits that can be used in SPICE. This means that we had two possibilities of comparing the simulated results with measurements. Either we could directly compare the scattering parameters or we could produce circuits and analyse them in SPICE and compare with measurements by translating the scattering parameters to total voltages and currents. For most of the comparisons presented in this report we have used the latter alternative since the goal of the project was to make engineering SPICE models.

The different steps in the approaches used in the project can be seen in Figure 3:1.
Figure 3.1. Methodology to characterise barriers.
4 Characterisation of barriers

4.1 Characterisation of barriers through measurements

As a basis for the generation of models and for the verification of models, measurements were performed on barriers. Depending on the different realisations of the barriers, different techniques were used to measure the barrier characteristics. Scattering-parameters have been used in many cases to describe a barrier, for instance for describing crosstalk between traces on a PCB and characteristics of surface mounted filters. The scattering parameter matrix represents the reflection and transmission coefficients in complex form, Figure 4:1. The scattering-parameters are measured using a vector network analyser.

![Figure 4:1. Transmission and reflection on a two-port.](image)

Measurements utilising a vector network analyser are made with a calibration technique where systematic errors due to mismatch, attenuation, delay etc. are removed by measurements on predefined calibration standards. Advanced vector network analysers offer alternative procedures to perform the calibration, e.g. using user-made calibration standards that are placed inside a fixture. After calibration the calibration plane can be moved by an offset to correct for the phase difference that occur for example when adapters are removed, see Figure 4:2.

![Figure 4:2. Vector Network Analyser with calibration planes and offset.](image)
4.1.1 S-parameters for a two-port

The reflection coefficients and the transmission coefficients of a network with two ports are also known as the scattering parameters for the two-port. The scattering parameters are a more systematic way of naming the coefficients and the scattering parameter matrix offers a complete mathematical description of a network that eases calculations on the network.

\[
\begin{align*}
S_{11} &= \frac{V_1^-}{V_1^+}_{V_1^+=0} \\
S_{22} &= \frac{V_2^-}{V_2^+}_{V_2^+=0} \\
S_{21} &= \frac{V_2^-}{V_1^+}_{V_2^+=0} \\
S_{12} &= \frac{V_1^-}{V_2^+}_{V_1^+=0}
\end{align*}
\]

Figure 4.3. Scattering parameters for a two-port.

The parameter \( S_{11} \) (reflection in port one) is defined as the voltage wave exiting port one \( V_1^- \) divided by the incident voltage wave on port one \( V_1^+ \) when the incident voltage wave on port two \( V_2^- \) has zero amplitude. In the same manner the parameter \( S_{22} \) (transmission from port one to port two) is defined as the voltage wave exiting port two \( V_2^- \) divided by the incident voltage wave on port one \( V_1^+ \) when the incident voltage wave on port two \( V_2^- \) has amplitude zero.

The reflection on port two and the backward transmission are defined likewise but with the port numbers exchanged. The four scattering parameters for the two-port network can be written in matrix form in the following way:

\[
\begin{bmatrix}
V_1^- \\
V_2^-
\end{bmatrix} =
\begin{bmatrix}
S_{11} & S_{12} \\
S_{21} & S_{22}
\end{bmatrix}
\begin{bmatrix}
V_1^+ \\
V_2^+
\end{bmatrix}
\]

4.1.2 Transfer impedance

The transfer impedance or surface transfer impedance is a way to describe the high frequency characteristics of an electromagnetic shield in terms of lumped or distributed circuit elements. Properties influencing the performance of an electromagnetic shield are the skin depth, geometrical shape among others. By determination of the transfer impedance these properties are modelled by a circuit element giving the corresponding electric field on the secondary shield surface for a certain current on the primary shield surface. This model are then very suitable for generation of a SPICE model making it possible to simulate the penetration of an electromagnetic shield in a circuit simulator.
Figure 4.4. Surface current on primary shield surface giving raise to electric field on secondary shield surface.

Measurement of transfer impedance is the standard procedure to characterise shielding efficiency of coaxial cables. A current is injected on the outside surface of the outer conductor. Due to leakage a voltage appear between the inner and the outer conductor of the coaxial cable. The leakage is modelled by the transfer impedance. The technique can also be used for characterising conductive gaskets and connector backshells.

\[ I \rightarrow V, U \]

Figure 4.5. Simplified sketch showing the measurement of transfer impedance in a coaxial cable shield.

4.1.3 Fixture for measuring transfer impedance for connectors

For the generation of SPICE models including the shielding effect of cable and connector backshell screens, it is necessary to describe the shielding efficiency in a way suitable for use in a SPICE simulator. By choosing a transfer impedance description this is accomplished and it also gives possibilities to perform measurements with a transfer impedance fixture. A useful fixture design was found in the literature [11].

Figure 4.6. Fixture described in [11] for transfer impedance measurements of shielded connectors.
The signal injected in the far end of the connector under test gives raise to a current on the outer surface of the backshell. The current returns through the chassis of the fixture and is measured with an inductive current probe. The current probe is used for verifying that the injected current mainly follows the desired path. After a calibration the current probe can also give an absolute value of the injected current, this method has not been used here. A voltage is measured between the center and outer conductor in the near end of the fixture. The input current and voltage measured across the cable connected to the connector under test gives the transfer impedance. By measuring the reflection coefficient on the input connector of the fixture (S11) the input current can be found from the input power $I_{in} = \frac{P_{in}}{1 + S_{11} \cdot 50}$, the transfer impedance can then easily be calculated as

$$Z_t = \frac{U_{in}}{I_{in}}.$$  

### 4.2 Characterisation of barriers through simulations

#### 4.2.1 S-parameters for a multiport

All multiports considered in the project have an equal number of input and output ports, thus they are all multiports with an even number of ports. A general multiport with 2N-ports that can represent any of the studied barriers is shown in Figure 4:7.

![Diagram of a 2N-port multiport](image)

**Figure 4:7.** Multiport with 2N ports.

In order to distinguish between the input and output ports we start the numbering with the input ports, i.e. the input ports will have numbers ranging from 1 to N and the output ports numbers from N+1 to 2N. It should be noted that the ports are often connected unsymmetrical, e.g. the lower nodes of the ports are connected to a common point (the ground point).

As for all multiports we have several different choices to describe its behaviour with a matrix relating the input and output quantities, either in terms of total voltages and currents or scattered. The choice here is in terms of the scattered voltages since these are the ones measured with a vector network analyser. The matrix relation becomes:
\[
\begin{bmatrix}
V_1^+ \\
V_2^+ \\
\vdots \\
V_{2N}^+
\end{bmatrix} =
\begin{bmatrix}
S_{11} & S_{12} & \cdots & S_{12N} \\
S_{21} & S_{22} & \cdots & S_{22N} \\
\vdots & \vdots & \ddots & \vdots \\
S_{2N1} & S_{2N2} & \cdots & S_{2N2N}
\end{bmatrix}
\begin{bmatrix}
V_1^- \\
V_2^- \\
\vdots \\
V_{2N}^-
\end{bmatrix}
\]

or in a more compact form as:

\[
\begin{bmatrix}
V_{\text{in}}^- \\
V_{\text{out}}^-
\end{bmatrix} =
\begin{bmatrix}
[S_{11}] & [S_{12}] & [V_{\text{in}}^+]
\end{bmatrix}
\begin{bmatrix}
[S_{21}] & [V_{\text{out}}^+]
\end{bmatrix}
\]

where:

\[
\begin{bmatrix}
V_{\text{in}}^+ \\
V_{\text{out}}^-
\end{bmatrix} =
\begin{bmatrix}
V_1^+ \\
\vdots \\
V_N^-
\end{bmatrix},
\begin{bmatrix}
V_{\text{in}}^- \\
V_{\text{out}}^+
\end{bmatrix} =
\begin{bmatrix}
V_1^- \\
\vdots \\
V_N^+
\end{bmatrix},
\begin{bmatrix}
[S_{11}] & \cdots & [S_{1N}]
\end{bmatrix} =
\begin{bmatrix}
S_{11} & \cdots & S_{1N}
\end{bmatrix}
\]

The plus signs in the above formulas means waves incident on the ports and the minus signs waves that are coming out of the ports. It should be noted that since we have an equal number of input and output ports the sub-matrices \([S_{11}], [S_{12}], [S_{21}]\) and \([S_{22}]\) are all square matrices with the same dimension (N by N).

The incident and reflected voltage waves are related to the total voltages as \(V_i = V_i^+ + V_i^-\) and associated current waves are related to the voltage waves through the characteristic impedance, \(Z_0\), in the measuring system (usually 50 Ohm). Thus, \(I_i = \frac{1}{Z_0} (V_i^+ - V_i^-)\).

This means that with the knowledge of the S-matrix we can determine the usual total voltages and currents as well.

### 4.2.2 3D-simulations

Today there are many electromagnetic simulation tools available to help designers calculate S-parameters for a given geometry. Next to measurement, 3D-simulation is in many cases the most accurate method to do this. There are, however, many factors to consider when doing a 3D electromagnetic simulation. Even with modern computers with ever increasing calculating speeds it is often necessary for the designer to partition the system before doing a simulation. It is not possible to do a full-wave electromagnetic simulation of, for example, a complete PC motherboard. To do that the designer has to know where on the board to do detailed simulations and where to use a more simplified model.

To be able to calculate the current distributions, and from that the S-parameters for the design, the simulator has to divide the layout in a mesh. There are two different approaches here, uniform or non-uniform meshing, shown in Figure 4:8.
Figure 4:8. Uniform (left) and non-uniform (right) meshing.

Obviously, non-uniform meshing yields higher accuracy but on the other hand increases calculation complexity. The biggest advantage of non-uniform meshing is that we can use high resolution where it's needed, i.e. corners and edges, and lower resolution on large uniform areas. In that way we can get both higher accuracy and fewer cells than with uniform meshing.

The meshing is the most critical process when doing 3D-simulations, and it's always a compromise between simulation time and accuracy. It takes a lot of experience to do the meshing in the smartest way and to be able to predict what accuracy we can expect from the simulation. Another disadvantage is that due to the high complexity of the software, 3D-simulators are generally very expensive.

4.2.3 2D-simulations

A large class of barriers can be viewed as being two-dimensional. For these types of barriers it is much more efficient to use a 2D-simulation tool than a 3D-tool. With two-dimensional it is here meant that the cross-section doesn't vary along the barrier. Example of two-dimensional barriers would be parallel traces on a printed circuit board and pins in a D-sub connector. Barriers that are nearly 2D can often be approximated by a number of cascaded 2D structures. We can e.g. approximate two oblique traces on a printed circuit board with a number of sections with parallel traces in a so-called staircase approximation.

The electrical characteristics of a two-dimensional barrier are described by its cross-section and the material properties. One approach to determine a circuit representation for the barrier, is to first determine the per-unit length parameters and then from these the circuit representation. This approach is based on that the barrier can be viewed as being a multi-conductor transmission line. Thus, the requirements are that the cross-section is uniform with an extent that is small compared to the wavelength. An example of a simple circuit representation for a barrier that can be modelled as a transmission line with three conductors (reference not counted) is shown in Figure 4:9.
Figure 4:9. Circuit representation for a short section of a three-wire transmission line.

Considering the circuit diagram in Figure 4:9, the following relations between the entries in the per-unit length capacitance and inductance matrices and the values of the circuit elements can be found.

\[
[C] = \begin{bmatrix}
\sum_{k=1}^{3} c_{1k} & -c_{12} & -c_{13} \\
-c_{21} & \sum_{k=1}^{3} c_{2k} & -c_{23} \\
-c_{31} & -c_{32} & \sum_{k=1}^{3} c_{3k}
\end{bmatrix}, \quad [L] = \begin{bmatrix}
l_{11} & l_{12} & l_{13} \\
l_{21} & l_{22} & l_{23} \\
l_{31} & l_{32} & l_{33}
\end{bmatrix}
\]

For the general case where the barrier can be modelled as a transmission line with \( N \) conductors the dimension of the matrices will be \( N \) by \( N \). Note that \( N \) is the number of conductors with the reference conductor not counted.

The per-unit length parameters \([L]\) and \([C]\) can be computed by using numerical methods such as the method of moments (MoM) [3], the finite element method (FEM) [4] etc. For some simple cases it is even possible to use analytical formulas. The method used in this project is the finite difference method (FDM) [5, Sec. 3], mainly because it is simple to implement and that it easily can handle complicated cross-sections with different materials.

To exemplify how to compute the per-unit length parameters for a 2D barrier, consider the simple printed circuit board shown in Figure 4:10.

Figure 4:10. Cross-section of a printed circuit board with two conductors.

For this example the dimension of the per-unit length capacitance and inductance matrices will be two by two, since we have two conductors and a common reference (the ground plane). For the general case the \( ij \)-element in the capacitance matrix can be determined by letting the potential on all conductors except the \( j \)-th be equal to zero and
evaluating the charge on the \(i\)th conductor, i.e. \(C_{ij} = \frac{Q_i}{V_j} \int_{V_{x = 0, x = j}}^{}\). Thus, in order to determine the capacitance matrix for the PCB in Figure 4:10, we have to solve the Laplace equation for the configuration two times with different boundary conditions. For the general case of \(N\)-conductors we have to solve the Laplace equation \(N\) times. The solution of the Laplace equation gives the potential distribution in the region and we can by applying Gauss' law determine the charge per unit-length on conductor \(i\) as: \(Q_i = -\int_{l_i} \epsilon \nabla V \cdot d\vec{n} d_l\), where \(l_i\) is a closed line around conductor \(i\), \(\hat{n}\) is an outward directed unit vector and \(V\) is the potential distribution. The inductance matrix can be computed by the knowledge of the capacitance matrix for the case when all material in the cross-section is free space, i.e. \([L] = \mu_0 \epsilon_0 \begin{bmatrix} C_0 \end{bmatrix}^T\) where \([C_0]\) is the capacitance matrix when all dielectric material in the cross-section is replaced by free space.

The remaining problem now is to determine the potential distribution, \(V\), by solving the Laplace equation. The solution can be found by starting with Maxwell's equations for the two-dimensional electrostatic case. By approximating the derivatives with finite differences we can quite easily write down the following relation between the potential in neighbouring nodes in the finite difference mesh that is shown in Figure 4:11.

\[
V_{i,j} = \frac{V_{i+1,j} (\epsilon_A + \epsilon_B)}{2(\epsilon_A + \epsilon_B + \epsilon_C + \epsilon_D)} + \frac{V_{i-1,j} (\epsilon_C + \epsilon_D)}{2(\epsilon_A + \epsilon_B + \epsilon_C + \epsilon_D)} + \\
\frac{V_{i,j+1} (\epsilon_B + \epsilon_C)}{2(\epsilon_A + \epsilon_B + \epsilon_C + \epsilon_D)} + \frac{V_{i,j-1} (\epsilon_A + \epsilon_D)}{2(\epsilon_A + \epsilon_B + \epsilon_C + \epsilon_D)}
\]

By giving all nodes in the cross-section an initial estimate, \(V_{i,j}^0\), and by scanning through the nodes by an iterative procedure we can determine the potential distribution in the whole region and thereby we are able to compute the per-unit length parameters.

![Figure 4:11. A part of the finite difference mesh used for solving Laplace equation.](image)

Based on the method for computing the per-unit length parameters described above a computer code called FD2D was developed. The code has a Windows user interface where the cross-section of the barrier easily can be defined by simply drawing it on the screen, see Figure 4:12.
Figure 4:12. User interface of the finite difference program, FD2D, for determination of per-unit length parameters. In the figure a coaxial line is analysed.

The FD2D program has been validated against several test cases and the agreement has been found to be good with previously published results. As an example of a simple validation the characteristic impedance for an air coaxial cable with an inner to outer conductor radius ratio of five was computed using a grid size of 200 by 200 nodes. The computed impedance was 96.85 Ω, which compares well to the exact value of 96.57 Ω.

From the computed per-unit length parameters the FD2D program can generate a representative SPICE circuit file if the user enters the length of the barrier (transmission line). The program can also compute the S-parameters for a given frequency range specified by the user.

The S-parameters are determined by first setting up the chain matrix for the analysed structure [8] and then using relations between the chain matrix and the scattering matrix. The chain or T-matrix relates the total voltages and currents at the input ports (ports 1...N, referring to Figure 4:7) to the voltages and currents at the output ports (ports N+1...2N). For the 2N-port in Figure 4:7 the T-matrix is given as:

\[
\begin{bmatrix}
V_1 \\
\vdots \\
V_N \\
I_1 \\
\vdots \\
I_N \\
\end{bmatrix}
= 
\begin{bmatrix}
T_{11} & T_{12} & \cdots & T_{12N} \\
T_{21} & T_{22} & \cdots & T_{22N} \\
\vdots & \vdots & \ddots & \vdots \\
T_{N1} & T_{N2} & \cdots & T_{NN} \\
\end{bmatrix}
\begin{bmatrix}
V_{N+1} \\
\vdots \\
V_{2N} \\
I_{N+1} \\
\vdots \\
I_{2N} \\
\end{bmatrix}
\]
or in a more compact form as:

\[
\begin{bmatrix}
V_1 \\
V_2 \\
\vdots \\
V_N
\end{bmatrix}
= \begin{bmatrix}
A & B \\
C & D
\end{bmatrix}
\begin{bmatrix}
V_2 \\
\vdots \\
V_N
\end{bmatrix}
- \begin{bmatrix}
V_1 \\
\vdots \\
V_N
\end{bmatrix}
\]

where:

\[
V_1 = \begin{bmatrix}
V_{11} \\
\vdots \\
V_{1N}
\end{bmatrix},
V_2 = \begin{bmatrix}
V_{N+1} \\
\vdots \\
V_{2N}
\end{bmatrix},
A = \begin{bmatrix}
T_{11} & \cdots & T_{1N} \\
\vdots & \ddots & \vdots \\
T_{N1} & \cdots & T_{NN}
\end{bmatrix}
\]

e etc.

If the 2N-port represents a multi-conductor transmission line with a length \( \Delta L \) the sub-
matrices in the T-matrix can be written as, [8]:

\[
A = [X] \cosh(\gamma \Delta L) [X]^{-1}
\]

\[
B = [X] \sinh(\gamma \Delta L) [X]^{-1} [Z_C]
\]

\[
C = [Y_C] X \sin(\gamma \Delta L) [X]^{-1}
\]

\[
D = [Y_C] X \cosh(\gamma \Delta L) [X]^{-1} [Z_C]
\]

(4:2)

Where the matrix \([X]\) is a matrix with the eigenvectors to the matrix product of the inductance and the capacitance matrices, \([\gamma]\) is related to the eigenvalues and \([Y_C], [Z_C]\)
are the characteristic admittance and impedance matrices, respectively. These matrices
can be determined by using the following relations:

\[
[X]^{-1} [L] C [X] = \begin{bmatrix} 1/v_i^2 \end{bmatrix}, \text{ diagonal matrix (eigenvalues)}
\]

(4:3)

\[
\gamma_i = \frac{j \omega}{v_i}
\]

\[
[Y_C] = [L]^{-1} [X]^{-1} [1/v_i] [X]^{-1},
[Z_C] = [Y_C]^{-1}
\]

(4:4)

Where the matrices \([L]\) and \([C]\) are the per-unit length inductance and capacitance
matrices.

By using the above equations we are now able to determine the T-matrix at the frequency
\((\omega = 2\pi f)\) of interest. On the other hand, the S-parameters for the 2N-port in Figure 4:7
are defined as:

\[
\begin{bmatrix}
V_1^- \\
V_2^- \\
\vdots \\
V_N^-
\end{bmatrix}
= \begin{bmatrix}
S_{11} & S_{12} & \cdots & S_{12N} \\
S_{21} & S_{22} & \cdots & S_{22N} \\
\vdots & \vdots & \ddots & \vdots \\
S_{N1} & S_{N2} & \cdots & S_{N2N}
\end{bmatrix}
\begin{bmatrix}
V_1^+ \\
V_2^+ \\
\vdots \\
V_N^+
\end{bmatrix}
\]

(4:7)
or in a more compact form as:

\[
\begin{bmatrix}
V_1^- \\
V_2^-
\end{bmatrix} = \begin{bmatrix}
[S_{11}] & [S_{12}] \\
[S_{21}] & [S_{22}]
\end{bmatrix} \begin{bmatrix}
V_1^+ \\
V_2^+
\end{bmatrix} 
\]  \hspace{1cm} (4.5)

where:

\[
\begin{bmatrix}
V_1^+ \\
V_2^+
\end{bmatrix} = \begin{bmatrix}
V_1^+ \\
\cdots \\
V_N^+
\end{bmatrix}, \quad \begin{bmatrix}
[S_{11}] \\
[S_{12}] \\
\cdots \\
[S_{22}]
\end{bmatrix} = \begin{bmatrix}
S_{11} & \cdots & S_{1N} \\
\cdots & \cdots & \cdots \\
S_{N1} & \cdots & S_{NN}
\end{bmatrix} \quad \text{etc.}
\]

The plus signs in the above formulas means waves incident on the ports and the minus signs waves that are coming out of the ports.

The amplitudes of the incident and reflected voltage waves are related to the total voltages as:

\[
V_i = V_i^+ + V_i^-
\]  \hspace{1cm} (4.6)

The associated current waves are related to the voltage waves through the characteristic impedance, \(Z_0\), in the measuring system (usually 50 Ohm).

\[
I_i = \frac{1}{Z_0} (V_i^+ - V_i^-)
\]  \hspace{1cm} (4.7)

Now, by using relations 4.6 and 4.7 in the definitions given in equations 4.1 and 4.5 we can, after some manipulations, write down the relations between the chain and scattering matrices as:

\[
[S_{11}] = \left\{ [1] + \left[ A \right] + \frac{1}{Z_0} \left[ B \right] \left( Z_0 [C] + [D] \right)^{-1} \left( [A] + \frac{1}{Z_0} [B] \right) \left( Z_0 [C] + [D] \right)^{-1} - [1] \right\}^{-1}
\]

\[
[S_{12}] = \left\{ [1] + \left[ A \right] + \frac{1}{Z_0} \left[ B \right] \left( Z_0 [C] + [D] \right)^{-1} \left( [A] + \frac{1}{Z_0} [B] \right) \right\}^{-1}
\]

\[
[S_{21}] = -2 \left\{ [1] + \left( Z_0 [C] + [D] \right)^{-1} \left( [A] + \frac{1}{Z_0} [B] \right) \right\}^{-1} \left( Z_0 [C] + [D] \right)^{-1}
\]

\[
[S_{22}] = -\left\{ [1] + \left( Z_0 [C] + [D] \right)^{-1} \left( [A] + \frac{1}{Z_0} [B] \right) \right\}^{-1} (Z_0 [C] + [D])^{-1}.
\]

\[
[S_{22}] = \left\{ [A] + \frac{1}{Z_0} [B] + (Z_0 [C] + [D])^{-1} \left( Z_0 [C] + [D] \right)^{-1}
\]

\[
\]  \hspace{1cm} (4.8)
It should be noted that since we are dealing with matrices the order of multiplication is important so that the relations (4:8) should be handled with care.

To sum-up, the steps that are needed to be taken in order to compute the S-parameters from the per unit-length parameters are:

1. use equation 4:3 to determine the diagonalization matrix, \( [x] \)
2. determine the characteristic admittance and impedance matrices by the use of equation 4:4
3. use equations 4:2 in order to compute the sub-matrices in the chain matrix
4. use the relations in 4:8 to finally determine the sub-matrices in the S-matrix
5. repeat from step 3 for all frequencies of interest

4.3 Generation of SPICE circuit based on measured S-parameters

For some types of barriers, i.e. physical barriers, measurement is the only possible way of characterizing the behaviour. For these cases a method by which it is possible to deduce a network from measured data has been developed. In this approach we take the measured S-parameters and compute the same for an assumed network. The next step is to seek for optimal values for components in the network so that a best fit, in some respect, is found. By this procedure the wanted network representing the measured device can be determined. Of course, the success of this method is only guaranteed if the assumed network actually can represent the device under consideration. Thus, a basic knowledge of circuit theory and some experience are required.

Since a measurement with a network analyser usually gives the S-parameters of a network it is natural to use these parameters as the basis for the comparison with the assumed network. Thus, we have to compute the S-parameters for the network. This can be done by first computing the chain matrix for the network and then convert to S-parameters. The relations between the S-parameters and the chain parameters can be found in section 4.2.3.

When the S-parameters have been determined for all frequencies of interest we have to compare the values with measured S-parameters. Figure 4:13 shows an example of such a comparison.
Figure 4:13. Example of computed and measured S-parameter for a barrier.

In order to find component values in the network that will make the network representing the measured device we minimize the average difference between measured and computed S-parameters, i.e. we minimize the function \( Q = \sum |\Delta_i(f)| \).

The method described above to determine a circuit representation outgoing from measured S-parameters was implemented in a computer code called S-Calc. The user interface of S-Calc is shown in Figure 4:14.

Figure 4:14. User interface of the S-Calc program for determination of circuit representation of a barrier outgoing from measured S-parameters.
4.4 Transforming S-parameters to total voltages

When using a circuit simulator such as SPICE or measuring on a circuit with an oscilloscope the observed quantities are the total voltages and currents. With a network analyser on the other hand the measured quantities are the scattered voltages. Thus, in order to be able to compare results obtained in either of the two ways we must be able to convert from one representation to the other.

In order to illustrate how we can compute the total voltages in a circuit if we know the scattering parameters, consider the circuit in Figure 4:15.

![Circuit diagram]

**Figure 4:15. Circuit for illustrating how to determine total voltages from S-parameters.**

We assume that we know the S-parameters for the filter in Figure 4:15. We also assume that we connect passive as well as active circuits to each port of the filter, as indicated in the figure. Since the filter has four ports we know from section 4.2.1 that the S-matrix for the filter can be written as:

\[
\begin{bmatrix}
V_1^- \\
V_2^- \\
V_3^- \\
V_4^-
\end{bmatrix} = 
\begin{bmatrix}
S_{11} & S_{12} & S_{13} & S_{14} \\
S_{21} & S_{22} & S_{23} & S_{24} \\
S_{31} & S_{32} & S_{33} & S_{34} \\
S_{41} & S_{42} & S_{43} & S_{44}
\end{bmatrix}
\begin{bmatrix}
V_1^+ \\
V_2^+ \\
V_3^+ \\
V_4^+
\end{bmatrix}
\]

For a single impedance element, \( Z_i \), we can determine the reflection coefficient, which is defined as \( \Gamma_i = \frac{V_i^-}{V_i^+} = \frac{Z_i - Z_0}{Z_i + Z_0} \), where \( Z_0 \) is the characteristic impedance of the measuring system (usually 50 Ohm). When we have a voltage source in series with the impedance element we have the following relation between the scattering voltages:

\[ V_i^- = U - \frac{Z_0}{Z_i + Z_0} + \Gamma_i V_i^+ \]

Now, since \( V^- \) for the connected impedance element is equal to \( V^+ \) for the filter, and vice versa, we can, for the circuit in Figure 4:15, modify the scattering matrix above as follows.
\[
\begin{bmatrix}
V_1^- \\
V_2^- \\
V_3^- \\
V_4^-
\end{bmatrix} = \begin{bmatrix}
S_{11} & S_{12} & S_{13} & S_{14} \\
S_{21} & S_{22} & S_{23} & S_{24} \\
S_{31} & S_{32} & S_{33} & S_{34} \\
S_{41} & S_{42} & S_{43} & S_{44}
\end{bmatrix} \begin{bmatrix}
U \frac{Z_0}{Z_1 + Z_0} + \Gamma V_i^- \\
\gamma_2 V_2^- \\
\gamma_3 V_3^- \\
\gamma_4 V_4^-
\end{bmatrix}
\]

By first inverting the scattering matrix and then operating on each row we are able to obtain the following expression:

\[
\begin{bmatrix}
U \frac{Z_0}{Z_1 + Z_0} \\
0 \\
0 \\
0
\end{bmatrix} = \begin{bmatrix}
S'_{11} - \Gamma_1 & S'_{12} & S'_{13} & S'_{14} \\
S'_{21} & S'_{22} - \Gamma_2 & S'_{23} & S'_{24} \\
S'_{31} & S'_{32} & S'_{33} - \Gamma_3 & S'_{34} \\
S'_{41} & S'_{42} & S'_{43} & S'_{44} - \Gamma_4
\end{bmatrix} \begin{bmatrix}
V_i^- \\
V_2^- \\
V_3^- \\
V_4^-
\end{bmatrix}
\]

In the above expression primed elements in the matrix means entries in the inverted original S-matrix, thus \([S]^{-1} = [S']\). By inverting the matrix again we can determine the \(V^-\) according to:

\[
\begin{bmatrix}
V_1^- \\
V_2^- \\
V_3^- \\
V_4^-
\end{bmatrix} = \begin{bmatrix}
S'_{11} - \Gamma_1 & S'_{12} & S'_{13} & S'_{14} \\
S'_{21} & S'_{22} - \Gamma_2 & S'_{23} & S'_{24} \\
S'_{31} & S'_{32} & S'_{33} - \Gamma_3 & S'_{34} \\
S'_{41} & S'_{42} & S'_{43} & S'_{44} - \Gamma_4
\end{bmatrix}^{-1} \begin{bmatrix}
U \frac{Z_0}{Z_1 + Z_0} \\
0 \\
0 \\
0
\end{bmatrix}
\]

Finally we can compute the total voltages as:

\[V_i = U \frac{Z_0}{Z_1 + Z_0} + V_i^- (1 + \Gamma_i)\]
\[V_i = V_i^- (1 + \Gamma_i); \ i = 2, 3, 4\]

It should be noted that even though the above method was demonstrated by using the example in Figure 4:15 the same method can be used for the general case. The described method for determining the total voltages has been implemented in the FD2D program. This means that the program also can be used for computing e.g. the crosstalk in terms of total voltages for traces on a printed circuit board.
4.4.1 Simplified transformation for two-port measurements

In order to compare the measurement results with the simulations, one must make sure that the same quantities are compared. With a network analyser, the scattering parameters of the network studied are measured. These are the relations between the voltage waves entering and leaving the network. In SPICE, the normal way of evaluating a network is to study the relation between the total voltages at the network ports. One way of solving this problem is to design special sub circuits in SPICE that evaluates the S-parameters in the schematic [9]. Another way is to convert the measured data into total voltages. A simplified procedure is here described for a two-port network measured with a network analyser. For a two-port, the voltage waves are defined as in Figure 4:16.

\[ \frac{V_2}{V_1} = \frac{V_2^+ + V_2^-}{V_1^+ + V_1^-} = \frac{V_2^+ \cdot (1 + S_{22}) + S_{21} V_1^+}{V_1^+ \cdot (1 + S_{11}) + S_{12} V_2^+} = (V_2^- = 0) = \frac{S_{21}}{1 + S_{11}} \] (4.9)}

Figure 4:16. Definition of voltage waves entering and leaving a two-port network.

Let us say we want to evaluate the total transmission through the network, that is the ratio between \( V_2 \) and \( V_1 \). By developing the expression for this ratio to an expression using the voltage waves we find a relation between the waves, modified by the scattering parameters. By using the fact that the measuring port 2 of the analyser is free from reflections, the expression is simplified as shown in (4.9). For a correct comparison between simulations and measurements, one need in this case only to correct the measurements by the factor \( 1 + S_{11} \).
5 Creation of SPICE circuit files

5.1 Creation of netlist

From the FD2D program developed in the project, an extraction of a netlist can be made. This is a file containing the values of the L- and C-matrices arranged in a form that can be read by a SPICE-program, a so called *.cir file. If required, longitudinal losses in form of lead resistances can be added. This output file is then used as a base for making sub circuit files, which can be connected to SPICE models. For this purpose, it is of importance that the circuit file does not include node 0. This node is defined as ground reference in the final netlist and cannot be used for other purposes.

The FD2D program generates the inductance and capacitance values per unit length for the cross-section simulated. In the netlist extraction, the absolute values are computed by specifying the total length of the segment. These values are then converted into a pi-section structure, as shown in Figure 7:4. Depending on the segment electrical length with respect to the wavelength of interest, the pi-section may need to be split into a number of sequential sections. As a rule of thumb, each section should have an electrical length not exceeding one tenth of the wavelength at the highest frequency of interest.

5.2 Creation of models

With the *.cir file as input, this file is converted into a sub circuit library (*.lib file). The file can then be imported to the Model Editor. An example of a converted and imported file is shown in Figure 5:1, where the proper sub circuit syntax is added to the beginning and the end of the *.cir file.

In the Model Editor, a set of related sub circuits can be collected into a combined library file, as shown in Figure 5:1. In the next step, an automatic creation of models is performed using a chosen template. The model library part (*.olb) can now be opened in Capture in the part editing mode, as shown in Figure 5:2. The automatic generation of the models result in generic pin names using the related node numbers in the corresponding sub circuit. These names are changed in the part-editing mode, followed by a subsequent change of the pin names in the Fspice Template call, by entering new pin names in the Part Properties settings dialog. After entering the name of the new library in the library path, in the settings mode, the model is now ready for use.
Figure 5:1. Importing the generated sub circuits into the Model Editor. The content of one sub circuit (PAT4P16) is displayed, showing the header of the file.

Figure 5:2. Editing the automatically generated models in Capture. The model PAT1P1 has been given new pin names.
5.3 SPICE simulations

The program used in this project is Orcad PSpice 9.0 (for simulations) and Orcad Capture (for creation of schematics) [10]. The procedures used for creation of the models refer to the tools accessible in the Orcad program structure. For other programs, however, similar procedures should be available.

The simulations have been made in the frequency domain using a logarithmic sweep from 150 kHz to 4 GHz with 101 calculated points per decade. Standard values of the tolerance parameters used by the program have been kept.
6 Verification of models

6.1 Test PCB for evaluating crosstalk between traces

In order to verify the models for barriers on printed circuit boards a PCB with parallel traces was manufactured. The purpose of this board was to compare the computed crosstalk that was obtained by using the combination of the FD2D program and SPICE with measurements done with a vector network analyser. The configuration was also used in order to compare computed S-parameters using a 3D simulation program and FD2D. The layout of the printed circuit board is shown in Figure 6:1.

![Figure 6:1. Layout of printed circuit board used for measurement of crosstalk between traces.](image)

One side of the printed circuit board in Figure 6:1 was a ground plane and the thickness of the substrate was 1.6 mm. The dielectric constant for the material in the substrate was 4.7. All traces were designed to have a characteristic impedance of 50Ω. The overall dimension of the circuit board was 160 by 100 mm.
6.2 Test PCB for evaluating filter components

For evaluating circuit models for physical barriers that were obtained by first measuring the S-parameters and then using the S-Calc program a special PCB was manufactured as shown in Figure 6:2. The physical barriers placed on the PCB were the following surface mounted filters.

- Filter L1: Murata BLM21B222S (series inductor, 2000 Ω @ 100 MHz)
- Filter C1: Murata NFM40R11C222 (capacitor, 2200 pF)
- Filter C2: Murata NFM61R30T472 (capacitor, 4700 pF)
- Filter C3: Murata NFM41P11C204 (capacitor, 0.2 μF)

On the circuit board special traces were dedicated for the calibration of the vector network analyser. These are the three traces on the left in Figure 6:2.

![Figure 6:2. Layout of printed circuit board used for measurement of surface mounted filter components.](image)

One side of the printed circuit board in Figure 6:2 was a ground plane and the thickness of the substrate was 1.6 mm. The dielectric constant for the material in the substrate was 4.7. All traces were designed to have a characteristic impedance of 50Ω. The overall dimension of the circuit board was 160 by 100 mm.
6.3 Test PCB with combined barriers

For the purpose of studying somewhat more complex systems containing several combined barriers the printed circuit board shown in Figure 6:3 was manufactured. The barriers on the PCB were of the same type as the barriers already studied individually by using the layouts shown in Figures 6:1 and 6:2.

By exciting at different pins in the D-sub connector that was mounted on the right edge of the PCB and by using different measuring points (SMA-connectors) different barriers could be chosen to be included in the measurements/simulations.

Figure 6:3. Layout of printed circuit board used for measurements of combined barriers.

One side of the printed circuit board in Figure 6:3 was a ground plane and the thickness of the substrate was 1.6 mm. The dielectric constant for the material in the substrate was 4.7. All traces were designed to have a characteristic impedance of 50Ω. The overall dimension of the circuit board was 160 by 100 mm.
7 Results

7.1 D-sub connector

7.1.1 Measurements

The crosstalk in a 9-pin D-sub connector was measured in different configurations by the use of a vector network analyser. The purpose here was to obtain measured data of crosstalk in configurations that also was used in SPICE simulations and thereby be able to verify the generated simulation model. The set-up is made on a wooden table and using two about 5 cm long RG58 coaxial cables from the calibration plane (N-type connectors) to the D-sub connectors. The outer conductor of the coaxial cable was connected to the d-sub connector by a 'pigtail'. Pins in the D-sub connector that are not used in a configuration are left open (not terminated).

![Diagram](image1)

**Figure 7:1.** First setup for crosstalk measurements on the D-sub connector.

A variation of the set-up was also used where a ground reference plane was inserted between the male and female of the D-sub connector. By this arrangement the field coupling from one side of the D-sub connector the other side was reduced, this coupling effect could then be examined.

![Diagram](image2)

**Figure 7:2.** Second setup for crosstalk measurements on the D-sub connector, utilizing a ground reference plane.

Three parameters were altered during measurements to analyse how the crosstalk in the D-sub connector is influenced by its usage:
• With or without ground reference plane
• The connector casing as return conductor or a pin as return conductor
• Termination of transmitting and receiving pin with \( R_c = 1 \Omega \), \( R_i = 50 \Omega \) or \( R_c = 1 \, \text{M}\Omega \).

![Graph](image)

**Figure 7.3.** Measured crosstalk in a D-sub connector with five different configurations.

The measurements used a logarithmic frequency sweep in the range 150 kHz to 1 GHz with 401 frequency points. As can be seen in figure 7.3 the ground reference plane did not influence much on the measured crosstalk. It can also be seen that if the metallic casing of the connector is not connected to the cable shield resonant conditions occur that gives poor repeatability at some frequencies.

### 7.1.2 Simulations

Two models of the D-sub connector were created: one full-scale model containing circuit elements related to all the pins, and one reduced model with only the transmission lines for pin 1 and pin 2 modelled. The content of the reduced model is displayed in Figure 7.4. In the simulation, intended to simulate the measurement set-up, the connector model was connected to a 1V AC source in combination with resistor elements as in Figure 7.5. The 50 \( \Omega \) resistors simulate the output and input impedance of the network simulator. The two impedances \( R_M \) and \( R_L \) were varied in the simulations. D-sub pins not connected in the measurements were in the simulation connected to ground via high impedance (100 M\( \Omega \)). The program requires this procedure, since it does not tolerate floating pins in the schematic.

![Schematic](image)

**Figure 7.4.** Outline of the reduced model for the D-sub connector
7.1.3 Comparison measured and simulated results

The total voltage far end crosstalk at pin 2 caused by the excitation of pin 2 was simulated using both the reduced and full-scale model. $R_{NB}$ and $R_L$ were both set to $R = 50 \, \Omega$. The result is displayed in Figure 7:6. As shown, the reduced model deviates from the measurements more than the full-scale model. For an exact simulation, the whole content of the connector must be modelled, and not just the actual pins of interest. The difference between the models and the measurement results at low frequencies (below 50 MHz) lie mainly in measurement noise.

![Figure 7:6. Comparison of simulation and measurement of the far end crosstalk at pin 2 with $R = 50 \, \Omega$ terminations.](image)

Simulations were also carried out with varying values of the terminating resistors. The results are displayed in Figure 7:7. For these cases, a conversion of the measurement values was performed according to Eq.(4.9). The simulation results compared well to the corresponding measurement values, except for the low impedance case ($R = 1 \, \Omega$). For this case, it proved to be difficult to apply the terminating resistors to the D-sub connector in the test set-up without introducing pigtail effects in the connections. When introducing a pigtail inductance of 2 nH in series with each termination, the simulation corresponded better with the measurements. For the high impedance case, the converted measurement values show a peak at high frequencies (approx. 900 MHz), which is a result of a
narrowband low reflection coefficient \( S_{11} \). This phenomenon is not reflected in the simulation, this effect presumably is an effect of the measurement set-up.

Attempts to simulate a connector with a floating casing relative to ground were also made. Connecting the ground references of the connector to ground via a 100 M\( \Omega \) resistor simulated this. However, the SPICE engine requires that all nodes shall have a low impedance DC-path to ground for the initial DC bias calculation. For the connector model, all leads except the ground reference are related to each other via the mutual inductance coupling. Since this ground had no low DC impedance connection to node zero in the schematics, the simulation did not converge.

Figure 7:7. Comparison of simulation and measurement of the far end crosstalk at pin 2 with highly varying values of the termination resistors. Measurement values are converted according to Eq. (4:9).
7.2 Shielded D-sub connector

7.2.1 Measurements and results

The measurements of transfer impedance on the D-sub connector were performed using a fixture described in literature [11]. The fixture does not offer a homogenous current distribution but it gives transfer impedance data that compares well with published values. Figure 7:8 shows the transfer impedance for a 9-pin D-sub connector when all pins are connected together and the current is concentrated towards one of the broader sides of the connector backshell (close to pins 6-9).

![Graph showing transfer impedance data](image)

Figure 7:8. Measured transfer impedance for the D-sub connector.

7.3 Shielded cable

7.3.1 Measurements

Measurements of the shielding efficiency of a RG58 coaxial cable was performed with the IEC 96-1 transfer impedance fixture. The fixture is more than one meter long, which causes its frequency range to be limited to below 30 MHz. The outer coaxial conductor in the fixture is short circuited in its far end and acts as the excitation of the outer surface of the tested cable screen. The inner coaxial conductor constitutes of the cable under test and it is terminated with its characteristic impedance.

The transmission through the fixture is measured using the network analyser as the parameter S_{21}. Then the transfer impedance can be calculated as [12]:

\[ Z_t = \text{abs}(S_{21}) \cdot 2 \cdot 1.4 \cdot 60 \cdot \ln \left( \frac{60}{d} \right) \]

where \( d \) is the outer diameter of the cable screen under test.

In figure 7:9 the measured transfer impedance is shown and it can be seen that there is a simple frequency dependence for frequencies below 30 MHz, but higher frequencies gives unexpected transfer impedance values for the RG58 cable.
7.4 Filters

7.4.1 Measurements

A complete scattering parameter matrix ($S_{11}$, $S_{21}$, $S_{12}$ and $S_{22}$) was measured for four different surface mounted filters. The measurement was performed with a network analyser using an optional calibration technique TRM, (Trough, Reflection, Match). For the measurement a test PCB (shown in Figures 7:10 and 6:2) was designed with pads for the four filters and the three calibration standards making it possible to calibrate to the pad of the filter component. The calibration standards were made as:

- Through: A 100 mm microstrip with a characteristic impedance of 50 $\Omega$ and SMA connector in both ends.
- Reflection: A SMA connector to a 50 mm microstrip and a via to ground (i.e. a short).
- Match: A SMA connector to a 50 mm microstrip in series with two parallel 100 $\Omega$ resistors to a via each.
Figure 7:10. Test PCB for vector network analyser calibration and measurements of surface mounted filter characteristics.

The filter S-parameter data was obtained in the frequency range 1 MHz to 4 GHz and the results for the insertion loss (S21) for the four different filters is shown in figure 7:11.

Figure 7:11. Measured insertion loss for the four different filters in this study.

7.4.2 Simulations

In the simulation of the filters here studied, discrete SPICE model elements were created using the values extracted from the S-Calc program and inserted directly in the schematic. Using the model, the insertion loss was simulated. The insertion loss is the quotient between the voltage at the filter position with and without the filter inserted, as shown in Figure 7:12.
7.4.2.1 Filter L1

The ferrite bead filter was simulated with three different models of increasing complexity, having a parallel connection of inductor, capacitor and resistor. The configuration of the full-scale model is shown in the schematic graph with the simulation set-up in Figure 7:13. The value of the inductance was first computed using the nominal impedance for the filter at the given frequency. The value was then fine-tuned using S-Calc.

Figure 7:13. Schematic for the simulation set-up for filter L1. Component values for the filter are $L_2 = 2.9 \, \mu H$, $C_2 = 1.15 \, pF$, and $R_6 = 2 \, k\Omega$. 
7.4.2.2 Filter C1

The feed thru capacitor C1 was modelled as a series connection of a capacitor, an inductance, and a resistor, see Figure 7:14.

Figure 7:14. Schematic for the simulation set-up for filter C1. Component values for the filter are C1 = 1900 pF, L4 = 240 pH, and R2 = 85 mΩ.

7.4.2.3 Filter C2

The T-type 3-terminal feed thru capacitor C2 was modelled as a T-filter including the intentional ferrite bead inductances in the terminals, plus the lead inductance and resistance related to the capacitor part, see Figure 7:15.

Figure 7:15. Schematic for the simulation set-up for filter C2. Component values for the filter are C1 = 5500 pF, L1=L3=500 pH, L4 = 189 pH, and R2 = 29 mΩ.
7.4.2.4 **Filter C3**

This capacitor was of the same type as filter C1, and had the same model configuration. This filter, however, had different component values as displayed in Figure 7:16.

![Filter C3 Schematic](image)

*Figure 7:16. Schematic for the simulation set-up for filter C3. Component values for the filter are C1 = 220 nF, L4 = 280 pH, and R2 = 5 mΩ.*

7.4.3 **Comparison measured and simulated results**

7.4.3.1 **Filter L1**

The results, compared with the measurements are displayed in Figure 7:17. The L-model gives of course no information on the resonance behaviour of the filter, while the LC-model predicts the resonance frequency with good accuracy. However, the Q value of the filter is greatly exaggerated but is better predicted using the full LCR-model.

The correlation between the measurements and simulations are surprisingly good considering the relative simplicity of the LCR-model. There is no frequency dependence in the inductance, which is a well-known property of ferrite materials. Moreover, the ohmic loss is also constant with frequency. The conclusion is that the complexity of the model is sufficient.
Figure 7:17. Comparison of simulation and measurement of insertion loss for filter L1.

7.4.3.2 Filter C1

The feed thru filter C1 was simulated using the set-up in Figure 7:14. The comparison of the simulation results and the measurements are displayed in Figure 7:18. The filter shows resonance behaviour well predicted by the model.

Figure 7:18. Comparison of simulation and measurement of insertion loss for filter C1.

7.4.3.3 Filter C2

The feed thru filter C2 was simulated using the set-up in Figure 7:15. The comparison of the simulation results and the measurements are displayed in Figure 7:19. The results show, that a model using the ideal values of the T-filter does not predict the measured resonance behaviour. In fact, this resonance is dependent on the values of the stray
components and is predicted by the full-scale model. The prediction of the resonance frequency is off by approximately 20 MHz.

![Graph](image)

**Figure 7:19.** Comparison of simulation and measurement of insertion loss for filter C2.

### 7.4.3.4 Filter C3

The feed thru filter C3 was simulated using the set-up in Figure 7:16. The comparison of the simulation results and the measurements are displayed in Figure 7:20.

![Graph](image)

**Figure 7:20.** Comparison of simulation and measurement of insertion loss for filter C3.
7.5  Crosstalk on printed circuit board

7.5.1  Measurements

A Printed Circuit Board (PCB) was designed and used for measurements of crosstalk between adjacent conductors. The measured data was used for verification of models made from theory based on the geometry. On the PCB four pairs of adjacent conductors was made with different spacing and as cascaded sections with different width and spacing, see Figure 7:21 (and Figure 6:1).

Figure 7:21. Test PCB for measurements of crosstalk between adjacent conductors.

On each pair of conductors three configurations were measured:

- Crosstalk to far end of adjacent conductor (as in Figure 7:21)
- Crosstalk to near end of adjacent conductor, the instrument connected to the left side of the PCB in Figure 7:21 (side a).
- Crosstalk to near end of adjacent conductor, the instrument connected to the right side of the PCB in Figure 7:21 (side b).

For all measurements the two ports not connected to the instrument was terminated with 50 Ω terminations. The measured configurations covers 10 of the total 16 S-parameters of the four-port made of one conductor pair. Since reciprocity can be assumed 12 S-parameters is known.
Figure 7.22. Measured data for two different far end crosstalk and for two different near end crosstalk.

As can be seen in Figure 7.22 there is a difference in far end crosstalk between pair 1 (top pair in Figure 7.21) and pair 3 (second from bottom in Figure 7.22). There is also a difference between near end crosstalk on side a and side b for pair 3 even though it is less significant.

7.5.2 Simulations

7.5.2.1 3D-simulations with commercial programs

To make a comparison between FD2D and a commercial 3D-simulation tool, IE3D from Zeland Software was used. It is a relatively new tool that has been widely accepted as a good simulation tool, using the method of moments [3] and a non-uniform mesh. It is available both for the Windows platform as well as Sun Solaris. The latest version at the time of this project, 6.0, was only available for Windows, which was the version, used for these simulations. The price at this time for version 6.0 is $22k. Four different twin-trace structures were simulated for transmission, reflection, far-end crosstalk and near-end crosstalk on all ports. This gives a total of 16 S-parameters from S11 to S44. The different structures are shown in Figure 7.23.
Figure 7.23. The layout part of IE3D.

All four structures were simulated with an infinite ground plane, which should be a good approximation in this case. The simulation parameters were:

- dielectric thickness: 1600 µm
- dielectric constant: 4.7
- dielectric loss tangent: 0.013
- metal layer thickness: 35 µm
- metal layer conductance: $4.9 \times 10^7$ S/m
- meshing frequency: 10 GHz
- meshing cells per wavelength: 20
- terminating impedance: 50
- simulated frequencies: 50 in the range 100 MHz – 10 GHz
- simulating time: ~10 min/frequency (30min for the one to the right)
- simulating hardware: Intel Pentium II 450 MHz, 768 Mbytes RAM

How the ports were numbered and a close-up of the meshed structure is shown in Figure 7.24.
Figure 7:24. The port numbering (left) and how the structures were meshed (right).

The computed S-parameters compared to the ones obtained with the developed 2D-program FD2D are shown in the Figures below. From the results it can be seen that the agreement is quite good for lower frequencies. For frequencies higher than a few GHz the deviation is larger and can probably be explained by radiation losses that are taken into account in the 3D-program but not in the 2D.

Figure 7:25. S11 and S21 for twin trace 1 (the left most in Figure 7:23).
Figure 7.26. S31 and S41 for twin trace 1 (the left most in Figure 7.23).

Figure 7.27. S11 and S21 for twin trace 2 (second from left in Figure 7.23).
Figure 7:28. S31 and S41 for twin trace 2 (second from left in Figure 7:23).

Figure 7:29. S11 and S21 for twin trace 3 (third from left in Figure 7:23).
Figure 7:30. S31 and S41 for twin trace 3 (third from left in Figure 7:23).

Figure 7:31. S21 and S31 for twin trace 4 (right most in Figure 7:23).
Figure 7.32. S33 and S43 for twin trace 4 (right most in Figure 7.23).

7.5.2.2 2D-simulations with program developed in project

Using the output netlist generated by the FD2D program, a SPICE model was created using the procedure described in section 5. For each twin trace configuration, four port models were designed in the form of cascaded pi-sections consisting of mutual capacitances and inductances. Series resistors were also added in all leads. These resistors, intended to reflect the low frequency copper losses, had very small values, which lead to numerical problems. The values were therefore increased to a generic value to overcome these difficulties. This problem is further discussed in section 8.1.

SPICE models were created having varying number of cascaded pi-sections. Based on the rule of thumb that recommends having at least 10 pi-sections per wavelength in the transmission line, the lowest order of models had an expected upper frequency bound of approximately 150 MHz. The models having the highest order were expected to give good results up to 2.6 GHz.

The models were inserted in schematics reflecting the measurement set-up used in 7.5.1. One schematic is displayed in Figure 7.33. From these simulations, the far end and near end crosstalk was evaluated in terms of the quotient between the total voltages.

Figure 7.33. Simulation set-up for a twin trace pattern 3, 16pi-model, intended for crosstalk simulation.
7.5.3 Comparison measured and simulated results

Since the measurements were all carried out having 50 Ω terminations, no reflections occurred and thus no conversion of the measurement data had to be performed in order to make justifiable comparisons. Comparisons were made on both the far end and near end crosstalk.

7.5.3.1 Twin trace 1

The far-end crosstalk between the first pair of conductors were simulated using the three different models and compared to the measurements, see Figure 7:34. The simulation results all lie very close to the measurements in the low frequency region. As the frequency increases, the lower order models deviate from the measured data. The highest order model shows the right response for the whole frequency range.

The resonances occurring in the near-end crosstalk has a number of resonance dips as shown in Figure 7:35. The frequency value is well predicted by both the 4-sections and the 16-sections models for the first three, while the last one is only present in the results given by the 16-sections model.

![Graph](image)

Figure 7:34. Comparison of simulation and measurement of the far end crosstalk for twin traces 1 from trace a to trace b.
7.5.3.2 Twin trace 2

Since the second pair of traces has a discontinuity at the middle of the traces, a minimum of two pi-sections could be used. Results from the simulations and measurements on Twin trace 2 are displayed in Figure 7:36 and 7:37. Again, the simulated far-end crosstalk is closer to the measured and better predicted as the order of the models increase. The 2 pi-sections model correspond to measurement data up to approximately 300 MHz. Again, the resonance behaviour of the near end crosstalk is well predicted by the 16 pi-sections model in frequency but somewhat exaggerated in amplitude, while the 4 pi-sections model misses the highest resonance.
Figure 7:37. Comparison of simulation and measurement of the near end crosstalk for twin traces 2 from trace a to trace b.

7.5.3.3 Twin trace 3

Compared to pattern number two, the third twin trace had a larger discontinuity at the middle of the traces. For the same reason, the minimum number of pi-sections was two. The simulation results compared to measurement data are displayed in Figure 7:38 and 7:39. The results follow the same pattern as for Twin Traces 2.

Figure 7:38. Comparison of simulation and measurement of the far end crosstalk for twin traces 3 from trace a to trace b.
Figure 7.39. Comparison of simulation and measurement of the near end crosstalk for twin traces 3 from trace a to trace b.

7.5.3.4 Twin trace 4

The fourth pattern, having the centre half of the traces broadened, was modelled with a 4pi- and 16pi-sections. The simulation and measurement results are presented in Figure 7.40 and 7.41. For these cases, the results did not correspond well to the measurement data. The simulations for the near end crosstalk indicate some computation problems for the frequency region above 1 GHz.

Figure 7.40. Comparison of simulation and measurement of the far end crosstalk for twin traces 4 from trace a to trace b.
Figure 7:41. Comparison of simulation and measurement of the near end crosstalk for twin traces 4 from trace a to trace b.

7.6 Combined barriers for conducted emission and immunity

7.6.1 Measurements

In order to verify that the developed models behaves accurately when they are cascaded to form a more complicated circuit, measurements are performed on a Printed Circuit Board designed specially for this purpose, Figure 7:42 (and Figure 6:3).

Figure 7:42. PCB for verification measurements of cascaded barriers.
The connectors on the PCB is one 9-pin D-sub connector where four pins are connected to microstrips and the other five pins are not connected, there are also two SMA connectors for connecting the instrument to the PCB. On one of the traces a surface mounted feed through capacitor is used and the two microstrips that not are connected to SMA connectors are terminated with 50 Ω. All combinations of connections of network analyzer port 1 to the D-sub pin 6, 7, 8 and 9 and port 2 to one of the SMA connectors was performed. Examples of the obtained data are shown in Figure 7:43. The SMA connector that was not connected to the instrument was terminated with 50 Ω and left open in separate measurements. In Figure 7:43 examples are shown from the measured data.

![Graph showing signal levels at different frequencies](image)

**Figure 7:43. Example of verification measurements data.**

### 7.6.2 Simulations

A simulation set-up was created using the existing models for the D-sub connector and the filter. For the transmission lines, new models were created using the FD2D program. The transmission line models were created using 16 pi-sections for the triple trace, and 4 pi-sections for the single traces. As for the other trace models, the longitudinal resistors all had the generic value of 91 fΩ. The schematic representing the test PCB is displayed in Figure 7:44. All pins not connected in the measurements were drawn to ground via a 100 MΩ resistor in the simulation. In the same way as for the filter measurements, the insertion loss is simulated for the different set-ups.

![Schematic diagram of the test PCB](image)

**Figure 7:44. Schematics for the simulation set-up for the test PCB.**
7.6.3 Comparison measured and simulated results

Simulations were performed for a signal path entering the D-sub connector at pin 7 and leaving the PCB at the connector 1 (SMA1). A comparison with the corresponding measurement is displayed in Figure 7:45.

![Figure 7:45](chart1.png)

**Figure 7:45.** Comparison of simulations and measurement results for the test PCB. The excitation is made on pin 7 on the D-sub connector and the output is measured at SMA1.

To verify the correct behaviour of the triple TLM with respect to crosstalk, a simulation was performed for a signal path entering the PCB at the D-sub connector on pin 7 and leaving the board at connector 2 (SMA2). Comparison with corresponding measurement data is presented in Figure 7:46.

![Figure 7:46](chart2.png)

**Figure 7:46.** Comparison of simulations and measurement results for the test PCB. The excitation is made on pin 7 on the D-sub connector and the output is measured at SMA2.
7.7 Combined barriers for radiated emission and immunity

7.7.1 Measurements

For a verification of combined barrier models including coaxial cable and D-sub connector shielding efficiency, special measurements were performed. The measurement setup consisted of a RG58 coaxial cable connected with a shielded D-sub connector to the test PCB inside a shielded box, see Figure 7:47. By injection of a current into the outer surface of the cable screen the attenuation and phase shift of different signal paths can be measured at the two SMA connectors. The centre conductor of the coaxial cable was consequently connected to four pins of the D-sub connector (pin 6 to 9). The transmission was measured from the injected current to the voltage measured at the SMA connectors on the test PCB.

![Figure 7:47. Test PCB inside shielding box.](image_url)

An EM-clamp was used for current injection into the cable screen and the injected current was measured with a calibrated current probe. The entire measurement set-up is shown in Figure 7:48 (the current probe is not visible).

![Figure 7:48. Measurement set-up for combined barrier measurement including cable and connector shielding performance.](image_url)

All combinations of connections of the coaxial cable centre conductor to the D-sub pin 6, 7, 8 and 9 and one of the SMA connectors to the instrument was tested. As in previous verification measurements on cascaded barriers the SMA connectors not used were both left open and terminated with 50 Ω.
7.7.2 Simulations

To the circuit shown in Figure 7:33 circuit elements were added to model the transfer impedance of the coaxial cable shield and the transfer impedance of the D-sub connector backshell. In the simulation the circuit was excited by a current through the transfer impedance model giving rise to a voltage at pin 7 or pin 8 of the D-sub connector. The voltage at SMA 1 and SMA 2 was calculated and the ratios of these voltages to the excitation current were plotted. The simulation results for both connections of coaxial cable centre conductor to the D-sub connector are shown in Figure 7:49.

![Simulation graph](image)

**Figure 7:49.** Simulation results of system of combined barriers where the excitation current may represent an incident electromagnetic field.

The model for cable and connector shield leakage was based on the transfer impedance measurements of these parts. Because of the few measurements obtained and the limited upper frequency of the used transfer impedance fixtures the frequency interval of this simulation is chosen to 1 MHz to 60 MHz.

7.7.3 Comparison measured and simulated results

By plotting both measured values and simulated in the same diagram the combined models were verified. The agreement was found to be good in the case shown in Figure 7:50, where coaxial cable centre conductor was connected to D-sub connector pin 7 and voltage measured at SMA 2. Comparison for other configurations gave moderate agreement to poor agreement partly because of some measurements suffering from low signal to noise ratio at frequencies below 100 MHz.
Figure 7:50. Comparison of measurement and simulation on combined barriers including shielding leakage models based on transfer impedance data. Voltage at SMA 2 divided by current in cable shield, centre conductor connected to Pin 7.
8 Discussion - PSPICE accuracy

The SPICE models for the multiple transmission lines, such as the connector and the PCB traces, were formed as pi-sections. Besides the coupling capacitors and inductors, these also included longitudinal resistive elements representing the conductor DC-resistance in each lead, the ground plane included. When these components are inserted in the model, the computation matrix become unstable. Care should therefore be taken when selecting the values for these resistors. For the purposes of this project, where the barrier is mainly regarded as the interface between the conductors, the values of these resistors do not influence the crosstalk voltages. To overcome the difficulties with the models used here, we selected a generic value for all resistors in all models. The best performance was found for $R = 91 \, \Omega$. The simulation results for the 4 pi-sections model, having different values of the resistors, of twin trace 1 are compared with measurements in Figure 8.1. The agreement with measurement data varies with resistance in the low frequency region, while it does not influence the result for high frequencies (above 200 MHz). It should be noted, that irrespective of the number of pi-sections in corresponding models identical results are obtained in the low frequency region if they have the same resistor values. For that reason, it is not possible to distinguish between accurate and erroneous models only by comparing simulations generated by different model orders. In reality, the resistance increases with frequency. For future modelling, this dependency should be included for both the conductor and substrate losses at high frequencies. In the low frequency range, where the numerical problems occur, losses are not important and should therefore be disregarded.

Another aspect of the accuracy of the simulation is the need for a low impedance DC path to ground for all nodes. This may pose a problem when a connector having a floating casing is to be simulated. This could be the case when the connector is installed in a plastic enclosure. A way to solve this numerical problem could be to connect the casing to ground via a very high inductance, giving high frequency isolation. However, this may give rise to unexpected resonances, since the interior of the model contain capacitors with low capacitance values. The presumed resonance frequencies may then occur within the range of interest.

![Graph](image)

**Figure 8.1.** Comparison of simulation, using a 4 pi-sections model, and measurement of the far end crosstalk for twin traces 1 from trace a to trace b. The value of the resistive elements in the models is varied.
9 Conclusions

It is shown that a lumped circuit approach using SPICE is feasible for conducted EMC analysis of barriers. The generation of lumped circuit models is based on 2D and 3D field calculations for transmission like barriers, or circuit identification from measured data for barriers with unknown geometrical shape. SPICE simulations with generated lumped circuit models gave good agreement with measured data for frequencies up to 1 GHz. For models of transmission like barriers the upper frequency limit for good agreement is depending on the number of cascaded sections in the circuit model. The generated models based on circuit identification gave good agreement for frequencies up to 1 GHz and in specific cases up to 4 GHz, when simple networks were assumed. Better agreement in the high frequency range can of course be reached by assuming a more complex network.

If a product developing company is to implement the use of electrical simulations in its product development process, the simulations must give a substantial improvement of the design process. It is a well-known fact that late changes in the design are very costly and time consuming. If such modifications can be reduced by using simulation tools in the early stages of the design process, design costs and development time can drastically be reduced. However, if designers are to use such tools, they must be well adapted to their work situation and their use of other design tools. This project has focused on the development of models that can be used in ordinary existing simulation tools. The activities in the project are illustrated in Figure 9:1.

![Diagram](image)

**Figure 9:1. Illustration of the developed methods for generation of SPICE models for EMC barriers. The research areas where further work is most necessary are marked with numbers (not ordered by necessity).**

An obvious continuation of this work is the extension to barriers for radiated electromagnetic fields. To make the lumped circuit model generation possible in an engineering environment, it is desired to integrate and automate the process to reduce the required skill for making proper assumptions. This includes the development of
numerical field solver techniques applied to relevant input data, (1) in Figure 9:1. Additionally, it is necessary to develop techniques for generation of equivalent circuits by system identification on measured or manufacturer-supplied data, shown at (2) in Figure 9:1.

It is also of great importance to investigate the limitations and accuracy of the SPICE engine in this field of application ((4) in Figure 9:1) and to improve the lumped component matrix description by systematic methods ((3) in Figure 9:1).

The issues for further work mentioned above are all well in line with the research interest at SP, LTU and IVF.
10 References


[9] Orcad tech note 89, Beaverton, Oregon, USA.


Appendix 1  Project organisation

This project has lasted for two years and has been financially supported by Nutek. The partners have been SP, LTU, IVF, ACREO and ABB. All partners except ABB have got founding from Nutek. ABB's contribution to the project has been through own work and they have also manufactured the circuit boards that have been used for measurements. Jan Welinder, SP has been responsible towards Nutek and Jan Carlsson, SP has been the project leader.

In order to ensure that the line of work has been of interest for the industry a reference group with representatives from the industry was formed at the beginning of the project. This group has been present at meetings and their suggestions and comments have been appreciated.

The project organisation is shown in Figure A1.

Figure A1. Project organisation.