A Description of the Anisotropic Material Behaviour of Short Glass Fibre Reinforced Thermoplastics Using FEA

Hamid Ghazisaeidi

Department of Mechanical Engineering
Blekinge Institute of Technology
Karlskrona, Sweden
2006

A Description of the Anisotropic Material Behaviour of Short Glass Fibre Reinforced Thermoplastics Using FEA

Hamid Ghazisaeidi

Department of Mechanical Engineering
Blekinge Institute of Technology
Karlskrona, Sweden
2006

Thesis submitted for completion of Master of Science in Mechanical Engineering with emphasis on Structural Mechanics at the Department of Mechanical Engineering, Blekinge Institute of Technology, Karlskrona, Sweden.

Abstract:
The aim of this work is to investigate and present a theoretical method to describe the anisotropic material properties of short glass fibre reinforced thermoplastics.

In this work, the injection moulding process simulation in SIGMASOFT is combined with ABAQUS by a JAVA interface suggested, aiding a micromechanical model, in order to simulate a steady state response of short glass fibre reinforced thermoplastics under the application of a harmonic force.

Keywords:
Anisotropic, Material properties, Injection moulding, SIGMASOFT, ABAQUS, JAVA interface, Micromechanical model
Acknowledgements

This work was carried out at the institute of plastic processing (IKV), RWTH-Aachen, Germany, under the supervision of Dr. Claes Hedberg from Blekinge Institute of Technology (BTH), Prof. Dr.-Ing. Ernst Schmachtenberg and Dipl-Ing. Tim Arping from Rheinisch Westfälische Technische Hochschule Aachen (RWTH-Aachen).

I wish to express my sincere appreciation to Dipl-Ing. Tim Arping and for his guidance and professional engagement throughout the work at the institute of plastic processing (IKV) RWTH-Aachen. At the department of mechanical engineering, Blekinge Institute of Technology (BTH), I wish to thank Dr. Claes Hedberg for his valuable supports and suggestions.

Finally, I would like to thank Dipl-Ing. Marcel Brandt for his support during the work.

Karlskrona, January 2006

Hamid Ghazisaeidi

[Signature]
Contents

1 Notation 6
2 Introduction, Motivation and Objectives 10
3 Theory 13
   3.1 Short Glass Fibre Reinforced Thermoplastics 13
      3.1.1 Classification of Composite Materials 13
      3.1.2 Fibre Reinforced Composites 15
      3.1.3 Thermoplastics 15
      3.1.4 Injection Moulding (IM) 16
   3.2 Anisotropy 17
      3.2.1 Equations of General Anisotropic Elasticity 17
      3.2.2 Micromechanical Model 18
   3.3 Viscoelastic Behavior of Plastics 21
      3.3.1 Application of Viscoelasticity to Describe the Behavior of Plastics 22
   3.4 Dynamic Mechanical Analysis (DMA) 23
      3.4.1 Sinusoidal Oscillatory Test 24
   3.5 Acoustic Analysis 25
      3.5.1 Formulation of the Equations of Motion 26
      3.5.2 Hamilton’s Principle 27
      3.5.3 Lagrange’s Equations 27
      3.5.4 Single Degree of Freedom System (SDOF) 28
      3.5.5 Multiple Degree of Freedom System (MDOF) 28
   3.6 Introduction to Finite Element Method (FEM) 30
   3.7 Introduction to ABAQUS 31
   3.8 Dynamic Response Analysis in ABAQUS 32
      3.8.1 Step1 Natural Frequency Extraction 32
      3.8.2 Step2 Direct Solution Steady State Harmonic Response Analysis 32
   3.9 Material Behavior Description 33
      3.9.1 Frequency Domain Viscoelasticity 34
      3.9.2 Material Damping (Rayleigh damping) 35
   3.10 Introduction to SIGMASOFT 35
4 Simulations 37
   4.1 Acoustic Simulation of a Viscoelastic Model Plate 37
      4.1.1 Preprocessing 37
4.1.2 Postprocessing and Discussion
4.2 Acoustic Simulation of Short Glass Fibre Reinforced Model Plates with Different Fibre Orientations
   4.2.1 Introduction
   4.2.2 Implementation into Software
   4.2.3 Part Geometry in SOLIDWORKS
   4.2.4 Injection Moulding Simulation in SIGMASOFT
      4.2.4.1 Preprocessing
      4.2.4.2 Postprocessing
      4.2.4.3 Result Extraction (SIGMAlink)
   4.2.5 JAVA Interface
   4.2.6 Acoustic Simulation in ABAQUS
      4.2.6.1 Preprocessing
      4.2.6.2 Postprocessing and Discussion

5 Conclusion and Further Works

6 References
Appendices

A. ABAQUS input file for the polyamide model plate 75
B. ABAQUS input file for the PP GF 30 model plate 1 80
C. ABAQUS input file for the PP GF 30 model plate 2 85
D. Material Data in SIGMASOFT for the PP GF 30 model plates 90
1 Notation

\( A \) \hspace{1cm} \text{Amplitude of the harmonic displacement}
\( c \) \hspace{1cm} \text{Damping matrix}
\( c \) \hspace{1cm} \text{Damping coefficient}
\( c_c \) \hspace{1cm} \text{Critical damping coefficient}
\( C_{ijkl} \) \hspace{1cm} \text{Elastic modulus tensor}
\( E \) \hspace{1cm} \text{Elastic modulus}
\( E_s \) \hspace{1cm} \text{Storage elastic modulus}
\( E_l \) \hspace{1cm} \text{Loss elastic modulus}
\( f \) \hspace{1cm} \text{Frequency}
\( F \) \hspace{1cm} \text{Force matrix}
\( F \) \hspace{1cm} \text{Force}
\( g(t) \) \hspace{1cm} \text{Non-dimensional shear relaxation function}
\( G^* \) \hspace{1cm} \text{Complex shear modulus}
\( G \) \hspace{1cm} \text{Shear modulus}
\( G_s \) \hspace{1cm} \text{Storage shear modulus}
\( G_l \) \hspace{1cm} \text{Loss shear modulus}
\( G_\infty \) \hspace{1cm} \text{Long term shear modulus}
\( k \) \hspace{1cm} \text{Stiffness matrix}
\( k \) \hspace{1cm} \text{Stiffness constant}
\( K^* \) \hspace{1cm} \text{Complex bulk modulus}
\( K \) \hspace{1cm} \text{Bulk modulus}
\( K_s \) \hspace{1cm} \text{Storage bulk modulus}
\( K_l \) \hspace{1cm} \text{Loss bulk modulus}
<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>$K_{\infty}$</td>
<td>Long term bulk modulus</td>
</tr>
<tr>
<td>$m$</td>
<td>Mass matrix</td>
</tr>
<tr>
<td>$m$</td>
<td>Mass</td>
</tr>
<tr>
<td>$t$</td>
<td>Time</td>
</tr>
<tr>
<td>$T$</td>
<td>Kinetic Energy</td>
</tr>
<tr>
<td>$T$</td>
<td>Period</td>
</tr>
<tr>
<td>$u$</td>
<td>Displacement</td>
</tr>
<tr>
<td>$\mathbf{u}$</td>
<td>Displacement vector</td>
</tr>
<tr>
<td>$\mathbf{u}$</td>
<td>Velocity vector</td>
</tr>
<tr>
<td>$\ddot{\mathbf{u}}$</td>
<td>Acceleration vector</td>
</tr>
<tr>
<td>$U$</td>
<td>Stiffness energy</td>
</tr>
<tr>
<td>$V$</td>
<td>Elastic strain energy</td>
</tr>
<tr>
<td>$W_{nc}$</td>
<td>Work done by non-conservative forces</td>
</tr>
<tr>
<td>$\nu$</td>
<td>Poisson’s ratio</td>
</tr>
<tr>
<td>$\omega$</td>
<td>Undamped angular natural frequency</td>
</tr>
<tr>
<td>$\varpi$</td>
<td>Damped angular natural frequency</td>
</tr>
<tr>
<td>$\tau$</td>
<td>Shear stress</td>
</tr>
<tr>
<td>$\phi$</td>
<td>Phase angle</td>
</tr>
<tr>
<td>$\phi$</td>
<td>Eigenvector</td>
</tr>
<tr>
<td>$\gamma$</td>
<td>Shear strain</td>
</tr>
<tr>
<td>$\sigma$</td>
<td>Stress</td>
</tr>
<tr>
<td>$\delta$</td>
<td>Phase lag angle</td>
</tr>
<tr>
<td>$\varepsilon$</td>
<td>Strain</td>
</tr>
<tr>
<td>$\xi$</td>
<td>Measure of reinforcement geometry</td>
</tr>
<tr>
<td>$\zeta$</td>
<td>Damping factor</td>
</tr>
<tr>
<td>$\Phi_f$</td>
<td>Fibre volume fraction</td>
</tr>
</tbody>
</table>
\eta \quad \text{Viscosity}
\alpha_R \quad \text{Mass proportional Rayleigh damping factor}
\beta_R \quad \text{Stiffness proportional Rayleigh damping factor}

\textbf{Indices}

c \quad \text{Composite}
f \quad \text{Fibre}
m \quad \text{Matrix}
Abbreviations

CAE  Computer Aided Engineering
DMA  Dynamic Mechanical Analysis
DOF  Degree of Freedom
FEA  Finite Element Analysis
FEM  Finite Element Method
IM   Injection Moulding
MDOF Multiple Degree of Freedom System
PP   Polypropylene
PP GF 30 Thirty Percent Short Glass Fibre Reinforced Polypropylene
SDOF Single Degree of Freedom System
2 Introduction, Motivation and Objectives

Especially in the automotive field multitude of technical plastics parts are used today. Since, the rising demands of the customers the technical plastic parts have to also serve acoustic assignments, in addition to their usual function. Designing these plastic parts already at an early stage of development information about the acoustic behavior is needed.

Due their higher stiffness and strength short fibre reinforced thermoplastics in contrast to unreinforced plastics may also be established for applications that were particularly realized in metallic materials so far, e.g. applications in automotive industry (Figure 2.1). Besides the low specific weight their main advantage is cost-efficient large-lot production even of complex shaped parts with high functionality by injection moulding.

![Figure 2.1. Examples of complex technical injection moulded parts made of short fibre reinforced thermoplastics.](image)

The prediction of acoustic behavior of short fibre reinforced plastics is supposed to be strongly dependent on the orientation of the glass fibres; since there are major differences in the material properties in the fibre direction and transverse to it (Figure 2.2). The orientation of the glass fibres mainly results from the complex melt flow during filling of the mould
cavity. In addition to different local fibre orientation over the part fibre orientation also varies in direction of wall thickness (Figure 2.3).

Figure 2.2. Dependency of the mechanical behaviour on fibre orientation [10].

Figure 2.3. Development of fibre orientation due to flow processes [10].
The aim of the studies in this thesis is to predict the effects of anisotropic material properties of the short glass fibre reinforced thermoplastics on the acoustic part behavior. While doing this, the simulation of the acoustic behavior should be improved on the knowledge of fibre orientation that is completely dependent on the injection moulding process situations. Regarding the acoustic FE-simulation of the plastic parts the established characteristic values have to be placed in a precise way.

Therefore in this study, by using FEM, the effect of the fibre reinforcement on the steady state dynamic responses of the plastic model plates under a harmonic excitation is investigated. Plastic model plates of PP and PP GF 30 with different fibre orientations are studied.

In the first part of the work, a simple polyamide plate is designed using SOLIDWORKS and meshed in ABAQUS (CAE), in order to study the linear viscoelastic material modeling. Then the natural frequencies and the steady state dynamic responses of the polyamide plate are extracted. The aim of this part is to investigate the linear viscoelastic material model in order to expand it to the modeling of anisotropic behavior in short glass fibre reinforced thermoplastics.

After that in the second simulations, the acoustic behavior of a PP plate and PP GF 30 model plates are studied for two months, where the motivation for this work is to show that how the injection moulding process simulation in SIGMASOFT can be combined to get the fibre orientation in order to put them into the ABAQUS calculation finding anisotropic behavior.
3 Theory

3.1 Short Glass Fibre Reinforced Thermoplastics

Fibre reinforced polymer composite systems have become increasingly important in a variety of engineering fields. The rapid growth in the use of composite materials in structures has required the development of structure mechanics for modelling the mechanical behaviour and the analysis of structural elements made of composite material as laminate or sandwich beams, plates, shells and injection moulded short glass fibre reinforced thermoplastic [1].

3.1.1 Classification of Composite Materials

Summarizing the aspects defining a composite as a mixture of two or more distinct constituents or phases it must be considered that all constituents have to be present in reasonable proportions that the constituent phases have quite different properties from the properties of the composite material. Man-made composites are produced by combining the constituents by various means. Figure 3.1 demonstrates typical examples of composite materials. Composite can be classified by their form and the distribution of their constituents (Figure 3.2). In the fibre reinforced plastics, the arrangement and the orientation of long or short fibres determine the mechanical properties of composites and the behavior ranges between a general anisotropy to an isotropy. Fibre reinforced composites are very important and in consequence this thesis work will essentially deal with modeling and analysis of elements composed of fibre reinforced material.

The most advanced composites are polymer matrix composites. They are characterized by relatively low costs, simple manufacturing and high strength. Their main drawbacks are the low working temperature, sometimes high coefficient of thermal and moisture expansion and, in certain directions, low stiffness. Polymer matrix composites are usually reinforced by fibres to improve such mechanical characteristics as stiffness, strength, etc. Fibres can be made of different materials (glass, carbon, aramid, etc). Glass fibres are widely used because their advantages include high strength, low costs and high chemical resistance [1].
Figure 3.1. Examples of composite materials with different forms of constituents and distributions of the reinforcements. a. Laminate with uni- or bidirectional layers, b. Irregular reinforcement with long fibre, c. Reinforcement with particles, d. Reinforcement with plate strapped particles, e. Random arrangement of continuous fibres, f. Irregular reinforcement with short fibres, g. Spatial reinforcement, h. Reinforcement with surface tissues as mats, woven fabrics, etc. [1].

![Diagram of composite materials with different forms of constituents and distributions of the reinforcements.]

Figure 3.2. Classification of composites [1].

The fibre length, their orientation and their material behaviors are main factors which contribute to the mechanical performance of a composite material. Although matrices by themselves generally have low mechanical properties (as) compared to fibres (except damping), they play following several important roles:
• Conduction of forces to fibres,
• Protection of fibres from environmental effects,
• Supporting of fibres under compression loads,
• Responsibility of damping in composite.

3.1.2 Fibre Reinforced Composites

The economic application of plastics materials to mass-produced precision engineered components has become possible largely as a result of the development of short fibre reinforced composites.

• Short fibre reinforced materials

Short fibre reinforced materials are the most commercially important materials and the ones that exhibit the most significant development and growth.

Some suppliers offer “long fibre” or “short fibres” reinforced thermoplastic material. As defined by long fibre compound suppliers materials contain fibre reinforcements that average 10 to 15 mm in length, while “short fibre” compounds contain fibre reinforcements that are something less than this, but average about 3 mm in length (Before injection moulding). After injection moulding process fibres length are about 0.1~1 mm [1].

• Glass Fibres

Reinforcements primarily give strength and stiffness to the composite. The predominant fibres used for reinforcement are made of glass. But there are a lot of kinds of fibres such as carbon (graphite), metal ceramic and etc.

Glass fibres are the most widely used form of reinforcement for plastic materials. Advantages of glass fibres over other reinforcements include a favorable cost/performance ratio with respect to dimensional stability, corrosion resistance, heat resistance, and ease of processing [1].

3.1.3 Thermoplastics

Thermoplastics are polymers that can be melted more than once as the molecular chains of these polymers do not crosslink and remain essentially unchanged (except for some shortening) each time they are processed. This
is in contrast to thermosets, which undergo a chemical reaction when processed, and hence cannot be remelted. Thermosets, however, generally have greater dimensional stability and heat and chemical resistance than thermoplastics. Paying attention to recycling issues; thermoplastics are superior compared to thermosets, for short fibre reinforced thermoplastics often are used as the matrix materials what needs need to an easy producibility in the injection moulding process [13].

3.1.4 Injection Moulding (IM)

Although many fabricating processes are employed to produce unreinforced plastic products, at least 50 % by weight of short fibre reinforced plastics go through injection moulding machines. For smaller lot quantities, however, other processes can compete with injection moulding (IM), among them being compression moulding, transfer moulding, casting, and thermoforming [2].

The IM process basically involves introducing the material into a cylindrical heating chamber by a rotating screw where the compound is melted under heat transfer and shearing. Then the melt is injected under high pressure into a matched metal cavity. The part then solidifies into its intended shape. With thermoplastic compounds, solidification occurs by cooling the polymeric melt, but with thermoset compounds, solidification occurs by heating the melt in the mould cavity to achieve polymerization and cross-linking. The next step involves ejecting the part from the mould. As shown in Figure 3.3, four basic mechanical units are combined to perform injection moulding: the mould, a plasticizing unit, a mould clamping unit and a control system unit [3].

![Diagram of Injection Moulding Process]

Figure 3.3. The elements of an injection moulding process [3].
3.2 Anisotropy

Composites are mostly anisotropic. The properties of anisotropic materials depend on the direction, whereas isotropic materials have no preferred direction. Short glass fibre reinforced composites are classified in anisotropic materials; therefore have some knowledge about anisotropy is necessary for analyzing their behaviors.

3.2.1 Equations of General Anisotropic Elasticity

The constitutive equation, Hooke's law, states that the stress is linearly proportional to strain [4],

\[ \sigma_{ij} = C_{ijkl} \varepsilon_{kl}, \]  

(3.1)

with \( C_{ijkl} \) as the elastic modulus tensor. There are 81 components of \( C_{ijkl} \), but taking into account the symmetry of the stress and strain tensors, only 36 of them are independent. If the elastic solid is describable by a strain energy function, the number of independent elastic constants is reduced to 21 by virtue of the resulting symmetry [15]

\[ C_{ijkl} = C_{klij}. \]  

(3.2)

This is the maximum number of independent elastic constants for any material symmetry. If the symmetry properties imposed by the microscopic nature of the material are considered, the number is typically less than 21. An elastic modulus tensor with 21 independent constants describes an anisotropic material with the most general type of anisotropy, triclinic symmetry [5]. Orthotropic symmetry is characterized by three mutually perpendicular mirror symmetry planes and twofold rotational symmetry axes perpendicular to these planes [4].
Materials with axisymmetry, also called transverse isotropy or hexagonal symmetry, are invariant to 60° rotation about an axis and describable by five independent elastic constants [9],

\[
\begin{bmatrix}
  C_{11} & C_{12} & C_{13} & 0 & 0 & 0 \\
  C_{22} & C_{23} & 0 & 0 & 0 & \\
  C_{33} & 0 & 0 & 0 & \\
  & C_{44} & 0 & 0 & \\
  & & sym. & C_{55} & 0 & \\
  & & & & C_{66} & \\
\end{bmatrix}
\]

(3.3)

with \( C_{44} = (C_{22} - C_{23}) / 2 \).

For example, a unidirectional fibre reinforced composite may have orthotropic symmetry if the fibres are arranged in a rectangular packing or transverse isotropy if the fibres are packed hexagonally [6]. Isotropic materials with properties independent of direction are described by two independent elastic constants [15].

### 3.2.2 Micromechanical Model

As mentioned, Hooke’s law written in tensor notation that describes the relation between stress and strain (Chapter 3.2.1) provides the basis for the calculation of the elastic behavior of plastics. If there are three orthogonal planes of symmetry in the material - as it can be found for short fibre reinforced thermoplastics - the material behavior is denoted orthotropic (orthogonal anisotropy) with nine elements in elasticity modulus tensor [10].
A comprehensive survey of different micromechanical models for the determination of the elastic properties of short fibre reinforced composites is given in [7]. At present from the multitude of published theories the equation sets of Tandon-Weng [8] and Halpin-Tsai [9] achieved the widest prevalence in commercial software.

The Halpin-Tsai equations are used in this thesis therefore they are explained in more details in the following.

- Halpin-Tsai Equations

The Halpin-Tsai equations are a simplified form of Hill’s generalized self-consistent model results with engineering approximation to make them suitable for the designing of the composite materials. Hill assumed a composite cylinder model in which the embedded phase considered continuous and perfectly aligned cylindrical fibres. Both materials were assumed to be homogeneous and elastically transversely isotropic (Eq. 3.4) about the fibre direction [9].

The Halpin-Tsai equations are tabulated in Table 3.1 for engineering constants where $E$, $G$ and $K$ represent elastic, shear and bulk modulus, respectively. $\xi$ is a measure of reinforcement geometry and also $\nu$ and $\Phi$ are Poisson ratio and fibre volume fraction, respectively. The indices $f$ and $m$ stand on fibre and matrix respectively for all the parameters in the Table 3.1.

Fibre volume fraction is calculated by Eq. 3.5;

$$ \Phi = \frac{V_F}{V_C} = \frac{A_F}{A_C} = \frac{1}{1 + \frac{1 - \Psi}{\Psi} \cdot \frac{\rho_f}{\rho_m}}, $$

(3.5)

where

- $\Psi = $ fibre weight fraction
- $V_F = $ volume of fibre
- $V_C = $ volume of composite
- $A_F = $ section area of fibre
- $A_C = $ section area of composite
\( \rho_f = \text{density of fibre} \)

\( \rho_m = \text{density of matrix} \).

<table>
<thead>
<tr>
<th>Table 3.1. Halpin-Tsai equations [9].</th>
</tr>
</thead>
<tbody>
<tr>
<td>( E_{11} = E_m \frac{1 + \xi \eta \Phi}{1 - \eta \Phi} )</td>
</tr>
<tr>
<td>( E_{22} = E_m \frac{1 + \xi \eta \Phi}{1 - \eta \Phi} )</td>
</tr>
<tr>
<td>( E_{33} = E_{22} )</td>
</tr>
<tr>
<td>( G_{12} = G_m \frac{1 + \xi \eta \Phi}{1 - \eta \Phi} )</td>
</tr>
<tr>
<td>( G_{12} = G_{13} )</td>
</tr>
<tr>
<td>( G_{23} = G_m \frac{1 + \xi \eta \Phi}{1 - \eta \Phi} )</td>
</tr>
</tbody>
</table>

In Table 3.1, \( l \) and \( d \) stand on the length and diameter of the fibre, respectively.

The relations between stiffness and engineering constants in Halpin-Tsai equations are tabulated in Table 3.2.
Table 3.2. Relation between stiffness tensor and engineering constants in Halpin-Tsai equations [9].

<table>
<thead>
<tr>
<th>$C_{ij}$</th>
<th>Engineering constants</th>
</tr>
</thead>
<tbody>
<tr>
<td>$(C_{22} + C_{23})/2$</td>
<td>$E_{22}$</td>
</tr>
<tr>
<td></td>
<td>$2(1 - \nu_{23} - 2\nu_{12}\nu_{21})$</td>
</tr>
<tr>
<td>$C_{12} = C_{13}$</td>
<td>$\frac{\nu_{21}E_{11}}{(1 - \nu_{23} - 2\nu_{12}\nu_{21})} = \frac{\nu_{12}E_{22}}{(1 - \nu_{23} - 2\nu_{12}\nu_{21})}$</td>
</tr>
<tr>
<td>$C_{11}$</td>
<td>$\frac{(1 - \nu_{23})E_{11}}{(1 - \nu_{23} - 2\nu_{12}\nu_{21})}$</td>
</tr>
<tr>
<td>$C_{44}$</td>
<td>$G_{23}$</td>
</tr>
<tr>
<td>$C_{55} = C_{66}$</td>
<td>$G_{12}$</td>
</tr>
<tr>
<td>$C_{12} / (C_{22} + C_{23})$</td>
<td>$\nu_{12}$</td>
</tr>
<tr>
<td>$C_{11}\left(\frac{2C_{12}^2}{C_{22} + C_{23}}\right)$</td>
<td>$E_{11}$</td>
</tr>
<tr>
<td>$\frac{[C_{11}(C_{22} + C_{23}) - 2C_{12}^2](C_{22} - C_{23})}{C_{12}C_{22} - C_{12}^2}$</td>
<td>$E_{22}$</td>
</tr>
</tbody>
</table>

3.3 Viscoelastic Behavior of Plastics

It is observed that a plastic, at specific temperature and molecular weight, may behave viscose as a liquid or solid depending on the speed (time scale) at which its molecules are deformed. This behavior, which ranges between liquid and solid, is generally referred to as the viscoelastic behavior or material response. If the deformations are small the modeling is linear viscoelastic. Non-linear viscoelasticity is required when modeling large deformations so that the material deformation is dependent on the grade of deformation. In this thesis work all simulations are assumed to be in the linear domain.

In linear viscoelasticity the stress relaxation test is often used, along with the time-temperature superposition principle [13] and the Boltzmann
superposition principle [12], to explain the behavior of plastics during deformation.

3.3.1 Application of Viscoelasticity to Describe the Behavior of Plastics

Most plastics exhibit a viscous as well as an elastic response to stress and strain. This puts them in the category of viscoelastic materials. Various combinations of elastic and viscous elements have been used to approximate the material behavior of plastics. Most models are combinations of springs and dash pots the most common one being the Maxwell model [12].

\[ \varepsilon = \varepsilon_e + \varepsilon_v. \]  \hspace{1cm} (3.6)

Similarly, the strain rates are written as

\[ \dot{\varepsilon} = \dot{\varepsilon}_e + \dot{\varepsilon}_v. \]  \hspace{1cm} (3.7)

Assuming the spring follows Hooke's law, the following relation holds

\[ \sigma = \varepsilon E. \]
\[ \dot{\varepsilon} = \frac{\dot{\sigma}}{E}. \]  

(3.8)

And the viscous portion, represented by the dash pot, is written as follows

\[ \dot{\varepsilon}_v = \frac{\sigma}{\eta}. \]  

(3.9)

Combining the last three equations result in

\[ \sigma = \frac{\eta}{E} \frac{d\sigma}{dt} = \eta \frac{d\varepsilon}{dt}, \]  

(3.10)

which is often referred to as the governing equation for the Maxwell’s model in differential form.

For more accurate estimate or realistic analysis the Maxwell model is not sufficient. For a better fit with experimental data it is common to use several spring-dash pot models in parallel. Such a configuration is often referred as a generalized Maxwell model [13].

### 3.4 Dynamic Mechanical Analysis (DMA)

The Dynamic Mechanical Analysis (DMA) is a very important apparatus in the modern polymer laboratory. DMA measures the mechanical properties of materials for different thermoplastics while they are subjected to a periodic stress, usually applied sinusoidally. An oscillating force is applied to a sample of the material and the resulting displacement of the sample is measured. From this the stiffness of the sample can be determined, and the sample modulus can be calculated. Measuring the time lag in the displacement compared to the applied sinusoidal force it is possible to determine the damping properties of the material.

Most polymeric materials show a combination of both types of behavior i.e. they react elastically, and flow to some extent at the same time (viscoelasticity). The stress and strain curves are therefore out of phase by an amount less than 90° [14].
3.4.1 Sinusoidal Oscillatory Test

In the sinusoidal oscillatory test, a specimen is excited with a low frequency stresses input which is recorded along with the strain responses. The shapes of the test specimen and the testing procedure vary significantly from test to test [13].

If the test specimen in a sinusoidal oscillatory test is perfectly elastic, the stress input and strain response would be as follows:

\[ \tau(t) = \tau_0 \cos \omega t , \]  
\[ \gamma(t) = \gamma_0 \cos \omega t . \] (3.11) (3.12)

For an ideally viscous test specimen, the strain response would lag \( \pi / 2 \) radians behind the stress input:

\[ \tau(t) = \tau_0 \cos \omega t , \]  
\[ \gamma(t) = \gamma_0 \cos(\omega t - \frac{\pi}{2}) . \] (3.13) (3.14)

Plastics behave somewhere in between the perfectly elastic and the perfectly viscous materials and their responses is described by

\[ \tau(t) = \tau_0 \cos \omega t , \]  
\[ \gamma(t) = \gamma_0 \cos(\omega t - \delta) . \] (3.15) (3.16)

The shear modulus takes a complex form of

\[ G^* = \frac{\tau(t)}{\gamma(t)} = \frac{\tau_0 e^{i\delta}}{\gamma_0} = \frac{\tau_0}{\gamma_0} (\cos \delta + i \sin \delta) = G_s + iG_l , \] (3.17)

which is graphically represented in Figure 3.5. \( G_s \) is usually referred to as storage shear modulus and \( G_l \) as loss shear modulus. The ratio of loss modulus to storage modulus is referred to as loss tangent or phase lag [13]:

\[ \tan \delta = \frac{G_l}{G_s} . \] (3.18)
3.5 Acoustic Analysis

As this thesis work is dealing with acoustic analysis, the vibration theory and finding appropriate method to set up the equations of motion is interesting.

Vibration theory is concerned with the oscillatory motion of the physical systems. The motions may be harmonic, periodic, or a general motion in which the amplitude varies with the time. When the deformation is quite small, the vibration is assumed linear.

Vibrations are the results of the combined effects of inertia and elastic forces. Inertia of the moving parts can be expressed in terms of the masses, moment of inertia, and the time derivatives of the displacements. Elastic restoring forces can be expressed in the terms of the displacements and stiffness of the elastic members.

While the effect of inertia and elastic forces tend to maintain oscillatory motion, the transient effect dies out because of energy dissipation. The process of energy dissipation is generally referred to as damping. Damping in general, has the effect of reducing the amplitude of vibration. Plastics materials are not perfectly elastic and they do exhibit damping, because of the internal friction due to the relative motion between internal molecular structures of the material during the deformation process. Such materials are referred to as viscoelastic solids as viscoelastic thermoplastics which are focused of this thesis work [16].
3.5.1 Formulation of the Equations of Motion

The success of mathematical analysis of vibration is dependent upon the correct formulation of the equations of motion and is based on some principles [17].

In order to obtain the equation of motion one may use the dynamic equilibrium (using Newton’s second law of motion) or the principle of virtual work (application of d’Alembert’s principle).

Considering the Newton’s second law for a dynamic system in equilibrium, the equation of motion can be written

\[ m\ddot{u} + c\dot{u} + ku = F. \]  (3.19)

In Figure 3.6 (for a single mass, spring damper system) the forces acting on the mass consist of the externally applied force \( F \), a restoring force \( ku \) due to the spring and a damping force \( c\dot{u} \) due to the viscous damper and an inertia force \( m\ddot{u} \).

For multi degree of freedom systems:

\[ \ddot{F}_i - \frac{d}{dt}\left[m_i\ddot{u}_i + c_i\dot{u}_i\right] = 0, \quad i = 1,2,\ldots,N \]  (3.20)

where considering a system of \( N \) masses and dampers [17].

![Figure 3.6. Single mass, spring, damper system [17].](image)

Since, the problem of vectorial addition of forces when the structure to be analysed is a complex one, this method is difficult to apply, therefore Hamilton principle or Lagrange’s equation (more convenient form for a discrete system) are more appropriate to use due to overcoming this problem.
3.5.2 Hamilton’s Principle

Although the principle of virtual displacement overcomes the problem of vectorial addition of forces, the virtual work itself is calculated from the scalar product of two vectors, one representing a force and one a virtual displacement. This disadvantage can be largely overcome by using Hamilton’s principle. [17]

The mathematical statement of Hamilton’s principle is

\[
\int_{t_1}^{t_2} \left( \delta(T - V) + \delta W_{nc} \right) dt = 0 ,
\]

where

\( T = \) Kinetic Energy of the system,

\( V = \) Elastic Strain Energy of the system,

\( W_{nc} = \) Work done by non-conservative forces.

The application of this principle leads directly to the equations of motion for any system. It can be applied to both discrete, multi-degree of freedom systems and continuous systems. The advantage of this freedom formulation is that it uses scalar energy quantities. Vector quantities may only be required in calculating the work done by the non-conservative forces.

3.5.3 Lagrange’s Equations

When Hamilton’s principle is applied to discrete systems it can be expressed in a more convenient form as follows:

\[
\frac{d}{dt} \left( \frac{\partial T}{\partial \dot{u}} \right) + \frac{\partial D}{\partial u} + \frac{\partial U}{\partial \dot{u}} = F ,
\]

where \( T = \frac{1}{2} mu^2 , U = \frac{1}{2} ku^2 , D = \frac{1}{2} c\dot{u}^2 . \)

Substituting this equation becomes same as the Eq. 3.19 [17].
3.5.4 Single Degree of Freedom System (SDOF)

In Eq. 3.19 with the assumptions of \( c = 0 \) and \( F(t) = 0 \), the equation becomes \( m\ddot{u} + ku = 0 \) (undamped free vibration). The solution of this equation is a pure time-harmonic oscillation and given by

\[
u(t) = A\sin(\omega t - \phi),
\]

where the amplitude \( A \) and phase \( \phi \) are determined by the initial conditions and \( \omega \) is the undamped natural frequency \( \omega = \sqrt{\frac{k}{m}} \). The solution describes an oscillatory motion without damping. With \( F(t) = 0 \) the equation becomes

\[
m\ddot{u} + cu + ku = 0.
\]

This is the standard form of the second order differential equation of motion that governs the linear vibration of damped single degree of freedom systems. The solution of Eq. 3.24 is given by

\[
u(t) = Ae^{-\xi\omega t} \sin(\omega t + \phi).
\]

where \( \xi \) is a dimensionless quantity. It is called as damping factor and defined as \( \xi = \frac{c}{c_c} \), while the critical damping coefficient \( c_c \) is defined as \( c_c = 2m\omega = 2\sqrt{km} \), and \( \omega \) is the system natural frequency defined as \( \omega = \sqrt{\frac{k}{m}} \). \( \sigma \) is the damped natural frequency \( \sigma = \omega \sqrt{1 - \xi^2} \).

The nature of the system’s motion depends on the value of \( \xi \) in Eq. 3.25 [18].

3.5.5 Multiple Degree of Freedom System (MDOF)

It is more convenient to use matrix notations to write the differential equations of systems, which have more than one degree of freedom. For a damped system with \( N \) degrees of freedom, the equation of motion can be written in matrix form as:
\[ m \ddot{u} + c \dot{u} + k u = F(t), \]  
\tag{3.26} 

where \( u, \dot{u} \) and \( \ddot{u} \) are the vectors of displacements, velocity and accelerations defined as:

\[
\begin{bmatrix}
  \mathbf{u} \\
  \mathbf{u}_1 \\
  \mathbf{u}_2 \\
  \vdots \\
  \mathbf{u}_N
\end{bmatrix}, \quad
\begin{bmatrix}
  \mathbf{\dot{u}} \\
  \mathbf{\dot{u}}_1 \\
  \mathbf{\dot{u}}_2 \\
  \vdots \\
  \mathbf{\dot{u}}_N
\end{bmatrix}, \quad \text{and} \quad
\begin{bmatrix}
  \mathbf{\ddot{u}} \\
  \mathbf{\ddot{u}}_1 \\
  \mathbf{\ddot{u}}_2 \\
  \vdots \\
  \mathbf{\ddot{u}}_N
\end{bmatrix},
\]

and \( \mathbf{m}, \mathbf{c} \) and \( \mathbf{k} \) are the mass matrix, damping matrix and stiffness matrix respectively and are given as:

\[
\mathbf{m} = \begin{bmatrix}
  m_{11} & m_{12} & \cdots & m_{1N} \\
  m_{21} & m_{22} & \cdots & m_{2N} \\
  \vdots & \vdots & \ddots & \vdots \\
  m_{N1} & m_{N2} & \cdots & m_{NN}
\end{bmatrix}, \quad \mathbf{c} = \begin{bmatrix}
  c_{11} & c_{12} & \cdots & c_{1N} \\
  c_{21} & c_{22} & \cdots & c_{2N} \\
  \vdots & \vdots & \ddots & \vdots \\
  c_{N1} & c_{N2} & \cdots & c_{NN}
\end{bmatrix}
\quad \text{and} \quad (3.27)
\]

\[
\mathbf{k} = \begin{bmatrix}
  k_{11} & k_{12} & \cdots & k_{1N} \\
  k_{21} & k_{22} & \cdots & k_{2N} \\
  \vdots & \vdots & \ddots & \vdots \\
  k_{N1} & k_{N2} & \cdots & k_{NN}
\end{bmatrix},
\]

The coefficients \( m_{ij} (i, j = 1,2,\ldots,N) \), \( c_{ij} (i, j = 1,2,\ldots,N) \) and \( k_{ij} (i, j = 1,2,\ldots,N) \) are termed mass, damping and stiffness matrix, respectively.

In the simplest theoretical case of \( N \)-DOF vibrations, there is no damping and no forcing (undamped free vibration), and the equations of motion are:

\[ m \dddot{u} + k u = 0. \]  
\tag{3.28} 

Assuming the solution to be time-harmonic \( \mathbf{u}(t) = \varphi \cos \omega t \), one obtains an eigenvalue problem for the determination of \( \omega \) and \( \varphi \)

\[ (\mathbf{k} - \omega^2 \mathbf{m}) \varphi = 0, \]  
\tag{3.29} 

\[ m, c, k \]
where $\omega^2$ is an eigenvalue and $\varphi$ is the associated eigenvector. For nontrivial solutions $\varphi \neq 0$ to exist, the determinant of the coefficient matrix must vanish, i.e.

$$\left| k - \omega^2 m \right| = 0.$$  

(3.30)

Expanding the determinant one obtains an n-degree polynomial in $\omega^2$, this solution called frequency-equation. The $N$ degrees of this polynomial provide a set of eigenvalues $\omega_i^2$, $i = 1, 2, ..., N$, and the corresponding values $\omega_i$ are the un-damped natural frequencies. Substituting each $\omega_i$ into Eq. 3.29 one obtains the associated eigenvectors $\varphi_i$, $i = 1, 2, ..., N$, also termed natural modes or mode shapes [18].

### 3.6 Introduction to Finite Element Method (FEM)

In the field of engineering design we come across many complex problems, the mathematical formulation of which is tedious and usually not possible by analytical methods. At such instants engineers often resort to the use of numerical techniques. Here lies the importance of FEM, which is a very powerful tool for getting the numerical solution of a wide range of engineering problems. The basic concept is that a body or structure may be divided into smaller elements of finite dimensions called as “Finite Elements”. The original body or structure is then considered as an assembly of these elements connected at a finite number of joints called as “Nodes” or “Nodal Points”. The properties of the elements are formulated and combined to obtain the properties of the entire body.

The equations of equilibrium for the entire structure or body are then obtained by combining the equilibrium equation of each element such that the continuity is ensured at each node. The necessary boundary conditions are then imposed and the equations of equilibrium are solved to obtain the variables required such as stress, strain, temperature distribution or velocity depending on the application.

Thus, instead of solving the problem for the entire structure or body in one operation, in the FEM, attention is mainly devoted to the formulation of properties of the constituent elements. A common procedure is adopted for combining the elements, solution of the equations and evaluation of the
variables required in all fields. Thus, the modular structure of the method is well exploited in various disciplines of engineering.

The method is not exact, but it can be very accurate if used wisely. The results, which are predicted by an experienced modeler, can be taken to be exact to engineering accuracy, being limited more by lack of precise knowledge of material properties, loads and the boundary condition than by the errors of numerical method. While interpreting the result of an FEA the engineer should be always aware of possible inaccuracy depending on these errors. The software ABAQUS is based on Finite Element Analysis to solve many engineering problems and is used in this thesis work.

### 3.7 Introduction to ABAQUS

ABAQUS is an engineering simulation program based on the finite element method, which can solve problems ranging from relatively simple linear analyses to the most challenging non-linear simulations. ABAQUS contains an extensive library of elements that can model virtually nearly, any geometry and most material properties. It has an equally extensive list of material models that can simulate the behavior of most typical engineering materials including metals, rubber, plastics, composites, reinforced concrete, crushable and resilient foams, and geotechnical materials such as soils and rock combined with the other material properties like density, specific heat capacity and etc. Designed as a general-purpose simulation tool, ABAQUS can be used to study more than just structural (stress-displacement) problems. It can simulate problems in such diverse areas as heat transfer, mass diffusion, thermal management of electrical components (coupled thermal-electrical analyses), acoustics, soil mechanics (coupled pore fluid-stress analyses), and piezoelectric analysis [19].

ABAQUS is simple to use even though it offers the user a wide range of capabilities. For example, problems with multiple components are modeled by associating the geometry defining each component with the appropriate material models. In most simulations, even highly nonlinear ones, the user needs only to provide the engineering data such as the geometry of the structure, its material behavior, its boundary conditions, and the loads applied to it. In a nonlinear analysis ABAQUS automatically chooses appropriate load increments and convergence tolerances. Not only does it choose the values for these parameters, it also continually adjusts them during the analysis to ensure that an accurate solution is obtained.
efficiently. The user rarely has to define parameters for controlling the numerical solution of the problem [19].

### 3.8 Dynamic Response Analysis in ABAQUS

ABAQUS offers several methods for performing dynamic analysis of problems which can be divided into two major groups: 1-Modal dynamics and 2-Complex harmonic oscillations.

All simulation in this thesis are dealing with two steps in which in the first step the natural frequency is extracted by Lanczos eigen-solver and the second step by providing a direct steady-state dynamic analysis, the responses of the system under harmonic excitation are determined.

#### 3.8.1 Step1 Natural Frequency Extraction

The natural frequencies of a system can be extracted using eigenvalue analysis (“Natural frequency extraction”) (See Chapter 3.5).

The frequency extraction procedure performs an eigenvalue extraction to calculate the natural frequencies and the corresponding mode shapes of a system. There are two methods to solve the eigenvalue matrix the Lanczos and the subspace iteration method. The Lanczos method is generally faster when a large number of eigenmodes is required for a system with many degrees of freedom. The subspace method is faster when only a few eigenmodes are needed.

The structural eigenvalue problem has received considerable attention since the advent of finite element models. Ramaswami [20] summarizes available methods for this problem. The most attractive one appears to be the Lanczos method that is used in the simulations of this thesis in order to find the natural frequencies.

#### 3.8.2 Step2 Direct Solution Steady State Harmonic Response Analysis

There are three ways of steady state analysis: Direct, Mode and Subspace based steady-state dynamic analysis. In the direct method the steady-state harmonic response is computed directly in terms of the physical degree of freedom of the model. The mode based analysis is based on modal
superposition techniques and the subspace analysis is based on a subspace projection method. For computing the steady-state response of simulations in this thesis the direct method is used, as it is more accurate than the other two methods especially when viscoelastic material behavior is present in the structure. However, it is more time consuming than the others.

The steady-state harmonic response of a system can be calculated in ABAQUS/Standard directly in terms of the physical degrees of freedom of the model ("Direct-solution steady-state dynamic analysis."). The solution is given in-phase (real) and out-of-phase (imaginary) components of the solution variables (displacement, velocity, stress, etc.) as functions of frequency. The main advantage of this method is that frequency-dependent effects (such as frequency-dependent damping) can be modelled. The direct method is the most powerful and accurate but also the most expensive steady state harmonic response procedure. The direct method can also be used if non-symmetric terms in the stiffness are important or if the model parameters depend on frequency.

As mentioned, steady-state dynamic analysis provides the steady-state amplitude and phase of the response of a system due to harmonic excitation at a given frequency. Usually such analysis is done as a frequency sweep by applying the loading at a series of different frequencies and recording the long term steady state response in ABAQUS/Standard the direct-solution steady-state dynamic procedure conducts this frequency sweep.

Therefore the direct steady-state dynamic analysis is chosen in the following problems;

- for nonsymmetric stiffness,
- when any form of damping other than modal damping must be included,
- when frequency dependent viscoelastic material properties must be taken into account.

3.9 Material Behavior Description

In order to describe the material behaviour (stiffness and damping) several methods are available. The most accurate method is using viscoelasticity theory and defines the mechanical behaviour of the plastic by data extracted from DMA.
3.9.1 Frequency Domain Viscoelasticity

A viscoelastic model can be used to specify frequency or time-dependent material behaviour. In the first simulation of this thesis work (Chapter 4.1) the material behavior is modeled by assuming a viscoelastic polyamide plate which is modeled in the frequency domain.

For modelling the viscoelastic behaviour of the plastic materials, the frequency domain modelling can be used. Frequency dependent shear and bulk moduli are used to include the dissipative part of the material behaviour.

- Determination of the viscoelastic material parameters

The dissipative part of the material behavior is defined by giving the real and imaginary parts of $g^*$ and $k^*$ as functions of frequency. Where $g^*(\omega)$ is the Fourier transform of the non-dimensional shear relaxation function $g(t) = \frac{G_R(t)}{G_\infty} - 1$. Expression for bulk relaxation function is similar to shear relaxation function. The moduli can be defined as functions of the frequency in the following three ways: by a power law, by tabular input or by a Prony series expression for the shear and bulk moduli [19]. In the first simulation of this thesis tabular frequency dependent data for input of material modeling is used.

- Tabular frequency dependence

The frequency domain response can alternatively be defined in tabular from on the data lines by giving the real and imaginary parts of $\omega g^*$ and $\omega k^*$, where $\omega$ is the circular frequency. The parameters are declared in the command line as:

$\omega \text{Re}(g^*), \omega \text{Im}(g^*), \omega \text{Re}(k^*), \omega \text{Im}(k^*)$ and $f$ (Hz)

where

$$\omega \text{Re}(g^*) = \frac{G_I}{G_\infty}, \quad \omega \text{Im}(g^*) = 1 - \frac{G_s}{G_\infty}, \quad \omega \text{Re}(k^*) = \frac{K_I}{K_\infty}, \quad (3.31)$$

$$\omega \text{Im}(k^*) = 1 - \frac{K_s}{K_\infty}, \quad G_\infty = G(t=\infty) \quad \text{and} \quad K_\infty = K(t=\infty)$$
This declaration was used to define the viscoelastic option in the ABAQUS program for analyzing the steady state response (Chapter 4.1).

3.9.2 Material Damping (Rayleigh damping)

In direct steady state dynamic analysis, it is very often to define the energy dissipation mechanisms (dashpots) as part of the basic model. ABAQUS provides “Rayleigh” damping for this purpose. Rayleigh damping can be used in direct-solution steady-state dynamic analyses to get quantitatively accurate results, especially near natural frequencies.

To define Rayleigh damping, two Rayleigh damping factors should be specified; $\alpha_R$ for mass proportional damping and $\beta_R$ for stiffness proportional damping. In general, damping is a material property specified as part of the material definition.

For a given mode $i$ the damping factor, $\xi_i$, can be expressed in terms of the damping factors $\alpha_R$ and $\beta_R$ as:

$$\xi_i = \frac{\alpha_R}{2\omega_i} + \frac{\beta_R \omega_i}{2},$$  \hspace{1cm} (3.32)

where $\omega_i$ is the natural frequency at this mode. This equation implies that, generally speaking, the mass proportional Rayleigh damping, $\alpha_R$, is damping for the lower frequencies and the stiffness proportional Rayleigh damping, $\beta_R$, damping at the higher frequencies.

The Rayleigh damping model is used in the following simulation of a short glass fibre reinforced thermoplastic plate (Chapter 4.2).

3.10 Introduction to SIGMASOFT

SIGMASOFT is a tool for the investigation of the filling and cooling/curing processes for thermoplastics, elastomers and thermoset materials. This program has an integrated geometry modeler. Modeling a running system in 3D provides the base for the numerical computations. A wide range of modeling functions allows also the preparation of geometries in the software itself. Furthermore, it facilitates accurate importing and
subsequent modification of CAD data via the different interface available. Automatic meshing of the geometry in SIGMASOFT is a further important key for rapid, accurate and flexible operation.

SIGMASOFT is based upon MAGMASOFT which is a simulation software for the metal casting process which has been used for more than ten years by more than 400 companies worldwide. With SIGMASOFT the user can optimize the part and the tool with respect to optimal part quality and cycle times.

SIGMASOFT is based on 3D volume elements. Parts with varying wall thickness can be simulated in a physical correct manner. Inserts and the tool are part of the model, both are calculated in 3D. This engineering program allows the calculation of filling, cooling and residual stresses in 3D volume elements also for fibre reinforced materials.

In this thesis work, SIGMASOFT was used to calculate the fibre orientation in the modeled part, which affect its mechanical behavior [21].
4 Simulations

4.1 Acoustic Simulation of a Viscoelastic Model Plate

The simulation of the acoustic behavior of a polyamide model plate was done in ABAQUS. The main goal of this task is the study of the material modeling by data from DMA in the frequency domain, and obtaining the result needed for comparison in the following study.

In order to simulate an isotropic model plate after defining the part geometry, the material was modeled by the frequency dependent data obtained from DMA measurements. The goal of this study is finding the behavior of the specimen as responses for displacement, velocity and acceleration with respect to frequency under the application of a 1 N amplitude harmonic force.

4.1.1 Preprocessing

In this stage the model of the physical problem was defined and an ABAQUS input file (Study_1, Appendix-A) was created.

- Part Geometry

The model plate was created in SOLIDWORKS, the CAE software, and imported as a “stl” file to ABAQUS. As it is shown in Figure 4.1, it is a plate in feature of 112.5 mm length, 110.5 mm width and 2 mm thickness in which two corners of the plate are filleted by a radius of 12 mm. this part geometry corresponds to an actual model geometry which is available at IKV that was the reason for choosing and modeling this model plate. In further works, it will be possible to carry out validation measurements for the simulation results of this part.
Material Modeling

In the property module the plate was termed as Polyamide which has a long term elastic modulus of 380 MPa, a density of 1.144 g/cm\(^3\) and a Poisson’s ratio of 0.28 and also the material was assumed to be isotropic and linear.

The chosen units were consistent and in SI (mm) form. The density of the material was necessary to define the mass matrix as eigenmodes and eigenfrequencies depend on it.

The main aim of the study is to get the steady state dynamic response of the model plate under harmonic excitation considering frequency dependent stiffness and damping behaviors of plastics. To get the damping effect in the response, viscoelasticity associated with the plastic model plate had been declared by using tabular frequency dependence of the material property as discussed in Chapter 3.9.

The frequency domain viscoelasticity was used to characterize the frequency dependent viscoelastic material properties, including the losses.
caused by the “viscose” (internal damping) nature of the polymer (plastic). The frequency domain viscoelastic material model describes the frequency-dependent material behavior for small steady-state harmonic oscillations. The frequency dependent data extracted from DMA measurements were used as frequency tabular data in order to complete the model plate material modeling.

From DMA measurements, the storage and loss elastic modulus ($E_s$ and $E_l$) can be derived for the interesting frequency interval (see Figure 4.2).

![Graph showing storage and loss elastic modulus](image)

*Figure 4.2. Storage and loss elastic modulus obtained from DMA.*

Using Eq. 4.1 and 4.2, the values of $G_s$, $G_l$, $K_s$ and $K_l$ are obtained:

$$G_s = \frac{E_s}{2(1 + v)} , \quad G_l = \frac{E_l}{2(1 + v)} ,$$  \hspace{1cm} (4.1)

$$K_s = \frac{E_s}{3(1 - 2v)} , \quad K_l = \frac{E_l}{3(1 - 2v)} .$$  \hspace{1cm} (4.2)

Figure 4.3 shows the frequency dependency of the storage and loss shear modulus Polyamide.
Considering Eq. 3.31; the parameters required, used as inputs in the frequency tabular data, $\omega \text{Re}(g^*)$, $\omega \text{Im}(g^*)$, $\omega \text{Re}(k^*)$ and $\omega \text{Im}(k^*)$ were calculated.

- Enmeshment and Element type

The model plate is not a complex geometry and does not have different structural elements. Because of this a single meshing for the whole part is possible. As the simulation is considering shear stress, the solid (continuum) element type was assigned to the part.

The linear hexahedral solid element type C3D8 is assigned to the model plate in the mesh module. The first letter of this element’s name indicates to continuum (solid) element family the element belongs.

In order to verify the element numbers, the first step (Natural frequency extraction) was performed 7 times with different element numbers. As Figure 4.4 shows, by increasing the element numbers; the first, second and third natural resonance frequencies were obtained more accurately. Due to going to quite suitable result by applying more than 2000 nodes; 2691 nodes were chosen for all simulations of this thesis.

Figure 4.3. Storage and loss shear modulus obtained from Eq. 4.1 and 4.2.
Figure 4.4. Accuracy in the result by increasing the element numbers.

- Steps involved in the modeling

Steps involved for the simulation were frequency extraction and direct steady state dynamic analysis.

In the first step the natural frequencies were calculated by using Lanczos eigensolver by assigning a frequency interval of 1 to 10000 Hz.

The direct steady-state dynamic analysis was assigned for the second step of this study in which provides the steady-state amplitude and phase of the response of a system due to harmonic excitation at a given frequency. The analysis was done as a frequency sweep by applying the loading at a series of different frequencies and recording the response. The frequency range of interest was selected from 1 to 10000 Hz and the number of frequencies at which results were required in this range was specified to 10000 on the data lines of the *STEADY STATE DYNAMICS command. The frequency spacing is linear.
• Load and boundary conditions

To define the boundary conditions one end of the plate was fixed and it was assigned with zero degrees of freedom by means of encastré option in the boundary condition command for all steps (Figure 4.5).

The concentrated harmonic load of 1 N amplitude was applied on the corner point of the plate which is highlighted in Figure 4.5. In the steady state dynamics step it is automatically considered as of sinusoidal nature and only the value of the amplitude has to be assigned. The frequency of the force is automatically adopted with the frequency sweep.

Figure 4.5. Load and boundary conditions.
4.1.2 Postprocessing and Discussion

In the first part of this section natural frequencies and associated modes of vibration are discussed, which give the deformation behavior of the system on corresponding eigenmode. While, in the analysis of the steady state dynamics response, the effect of the harmonic force on the displacement and velocity responses are investigated.

- Analysis done in step-1 (Frequency extraction)

The first four mode shapes of the natural frequency extraction step with corresponding undamped natural frequencies are shown in the Figure 4.6 and also the first ten undamped natural frequencies are tabulated in Table 4.1. The contour plots of the deformed plate show how the plate is deformed relatively. Figure 4.6 shows that opposite regions are more deformed than the regions near by the fixed end. And also there are other regions with zero deformation depending on the corresponding eigenform at each mode shape.

Table 4.1. Natural Resonance Frequencies (Hz).

<table>
<thead>
<tr>
<th>Mode No.</th>
<th>Frequencies (Hz)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>14.55</td>
</tr>
<tr>
<td>2</td>
<td>46.90</td>
</tr>
<tr>
<td>3</td>
<td>89.82</td>
</tr>
<tr>
<td>4</td>
<td>127.98</td>
</tr>
<tr>
<td>5</td>
<td>156.90</td>
</tr>
<tr>
<td>6</td>
<td>253.88</td>
</tr>
<tr>
<td>7</td>
<td>282.59</td>
</tr>
<tr>
<td>8</td>
<td>284.67</td>
</tr>
<tr>
<td>9</td>
<td>328.12</td>
</tr>
<tr>
<td>10</td>
<td>457.54</td>
</tr>
<tr>
<td>...</td>
<td>...</td>
</tr>
</tbody>
</table>
Analysis done in step-2 (Steady state dynamics)

In this analysis, beside the calculation of the natural frequencies also after the result values like the displacement, the velocity and the acceleration can be determined. In this section the steady state dynamics response of the polyamide model plate including material damping is achieved and the displacement, velocity and acceleration responses obtained at the node of the application of the harmonic force are extracted.

The first ten damped resonance frequencies are tabulated in Table 4.2, and the corresponding deformation of the model plate is shown in Figure 4.7 for the first four damped resonance frequencies.

**Figure 4.6. Deformation of the polyamide model plate at the undamped resonance frequencies.**
Table 4.2. Damped Resonance Frequencies (Hz).

<table>
<thead>
<tr>
<th>Mode No.</th>
<th>Frequencies (Hz)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>17</td>
</tr>
<tr>
<td>2</td>
<td>53</td>
</tr>
<tr>
<td>3</td>
<td>109</td>
</tr>
<tr>
<td>4</td>
<td>151</td>
</tr>
<tr>
<td>5</td>
<td>185</td>
</tr>
<tr>
<td>6</td>
<td>334</td>
</tr>
<tr>
<td>7</td>
<td>399</td>
</tr>
<tr>
<td>8</td>
<td>578</td>
</tr>
<tr>
<td>9</td>
<td>620</td>
</tr>
<tr>
<td>10</td>
<td>708</td>
</tr>
<tr>
<td>...</td>
<td>...</td>
</tr>
</tbody>
</table>

Regarding the material definition (Chapter 4.1), the stiffness value changes by frequency, so the stiffness value is not constant like the case of natural frequency extraction. From the comparison between the undamped natural frequencies and the corresponding damped ones; it is found that the damped resonance frequency values shifted from the undamped natural frequencies to the higher values. For example the first mode of undamped natural frequency is situated at 14.55 Hz, while in the damped case the first resonance peak lies on 17 Hz (comparison between Table 4.1 and 4.2). Because the stiffness value is considered constant in the first step of the simulation where the natural frequencies are extracted using the eigensolver, however considering viscoelastic frequency dependent material definition in the second step, the stiffness and damping values are changing corresponding to frequency increment. This proves the fact that any changing in material stiffness leads to higher or lower frequency shifting. It stands on this fact that the higher stiffness value leads to higher resonance frequencies and lower stiffness goes to lower resonance frequencies.

From the other point of view the effect of changing damping on the model plate response is performed at the peak value magnitudes.
Figure 4.7. Deformation of the polyamide model plate at the peak values of the damped frequencies.

Figure 4.8 illustrates the displacement, velocity and acceleration responses of the polyamide model plate to the harmonic force in a frequency interval between 1 to 1000 Hz. In this figure and in all three responses, the peaks are laid on the damped resonance frequencies and the effect of damping on the magnitude of the responses is based on the natural phenomenon of damping effects.
4.2 Acoustic Simulation of Short Glass Fibre Reinforced Model Plates with Different Fibre Orientations

4.2.1 Introduction

Basic requirement for a precise acoustic simulation of injection moulded parts made of short-fibre reinforced thermoplastics using FEA is information about the local fibre orientation in the part.

The properties of a unidirectional reinforced fibre/matrix-composite constitute the basis for the description of the local part stiffness of fibre reinforced plastics (Figure 4.9). They can be calculated by means of micromechanical models from the individual characteristics of fibre and matrix (e.g. Halpin-Tsai equations in Chapter 3.2.2). As input parameters the elastic properties of fibre and matrix, the fibre volume fraction and the
fibre geometry are needed (Table 3.1). On the other hand, the short fibres orient to a state between complete disorientation and perfect unidirectional alignment. Therefore, the stiffness properties have to be adjusted to the actual state of orientation.

Figure 4.9. Determination of the local stiffness of short fibre reinforced thermoplastics.

4.2.2 Implementation into Software

The complete procedure as it was realised at the Institute of Plastic Processing (IKV), Aachen, Germany, is shown in Figure 4.10. First of all a complete 3D-calculation of the injection moulding process is carried out using the SIGMASOFT package. In this context the geometry description of the part geometry description of the part may be a CAD-model (e.g. *.stl from SOLIDWORKS used in this thesis) or an adequate volumetric FE-mesh. The results of this simulation step, i.e. fibre orientation, are then mapped onto the FE-mesh for the acoustic analysis using the interface SIGMAlink (Part of SIGMASOFT). Finally a routine called,”3D-
SIGMAmeetsABAQUS” which is developed at the IKV is used to calculate the local anisotropic mechanical properties dependent on the local fibre orientation and the measured material data concerning stiffness.

Figure 4.10. Concept for the complete 3D-design of short fibre reinforced injection moulded parts.

To carry out the acoustic simulations ABAQUS/STANDARD was used as the FEA program. However, a special user subroutine is needed to realize the anisotropic stiffness analysis. Acoustic analyses were applied in ABAQUS for the model plates.
4.2.3 Part Geometry in SOLIDWORKS

Two models consisting of the part, runner, inlet and mould were designed in SOLIDWORKS. In the first model the runner is situated in the middle of the smaller side of the plate as a “fish tail distribution system” (Figure 4.11) while in the second model the melt runs to the mould by runner from one of the corners of the model plate (Figure 4.12). This diversity in the position of the runner leads to a very different melt flow in the models and therefore leads to different fibre orientations in the model plates after injection moulding.

(The models were saved in “stl” file format, which is a format readable in SIGMASOFT.)

![Figure 4.11. Model 1 designed in SOILDWORKS.](image)
4.2.4 Injection Moulding Simulation in SIGMASOFT

As it is mentioned; the information about the local fibre orientation in the part is the basic requirement for a precise acoustic design of injection moulded parts made of short-fibre reinforced thermoplastics using the FEA. In this section SIGMASOFT was used to simulate the injection moulding process.

4.2.4.1 Preprocessing

- Part Geometry

At first two projects were created, and then corresponding models, geometries were imported to SIGMASOFT as “stl” files. Then the different geometries were renamed to part, runner, inlet and mould. Afterwards the materials were assigned to the different geometries each project. Then the projects were ready for generating the meshes.
• Enmeshment

In the advanced tab of the mesh generation of the enmeshment module in SIGMASOFT, the plates were separated to the advanced meshing part because the results on the model plates are more important than for the mould, inlet or runner. Therefore the element number of the parts should be bigger compared to the rest of geometries. In order to have a fine enough enmeshment, the parameters for the accuracy, wall thickness, element size and smoothing in option part was filled by appropriate values. The meshed model plates with their runner and inlet parts are shown Figure 4.13 and Figure 4.14 for the model 1 and 2, respectively. The number of elements for the model plate 1 and 2 are about 278,000 and 168,000, respectively.

![Figure 4.13. The enmeshment stage in SIGMASOFT for the model 1. (278,264 part elements)](image-url)
• Simulation

The calculation can be started including the injection face of packing and cooling and with the preparation for fast postprocessing.

• Material definition

The material for the mould was selected from the standard library as steel with the temperature of 70 °C. In order to assign the material for the part, runner and inlet as a 30 % short glass fibre reinforced polypropylene (POLYFORT-FPP 30 GF-C, Schulman, Kerpen- Germany), a material was defined, due to the fact that this material is not available in the SIGMASOFT library.

With the aid of the databases CADMOULD and MC-BASE the data required for the material definition (30% Short Glass Fibre reinforced Polypropylene) was extracted. MC-BASE is a material data base program and CADMOULD is also an injection moulding simulation process program which has a vast material database. These parameters include rheological data, thermal properties and pvT properties which can be found in the Appendix-D.
After defining and assigning the material to respected parts, the models were ready to start the process simulation.

### 4.2.4.2 Postprocessing

After finishing the simulation the result can be checked in the postprocessing module. The most interesting result is the fibre orientation in the model plates that is not just unidirectional as the glass fibres are distributed in different local fibre orientations, even in the wall thickness direction.

The temperature filling stages are shown in Figure 4.15 for the model plate 1 and Figure 4.16 for the model plate 2.

![Temperature filling stages of the model 1.](image)
The fibre orientation for the model plate 1 is shown in Figure 4.17 and 4.18 in the middle and the outer layer of the wall thickness, respectively, similarly Figure 4.19 and 4.20 illustrate the fibre orientation for the model plate 2 in core and edge layers, respectively.

Figure 4.17. Fibre orientations in the core layer of the model plate 1.
Figure 4.18. Fibre orientations in the edge layer of the model plate 1.

Figure 4.19. Fibre orientations in the core layer of the model plate 2.
To have a better view Figures 4.21 and 4.22 show the section view of the model plate 1 in thickness direction for fibre orientations in 1 and 2 direction, respectively. Considering Figure 4.21, most fibres are not laid in 1 direction and there are some regions in the middle of the wall thickness that about 30 percent of fibres are turned in 1 direction.

Figure 4.22, proves this fact that most of the fibres in the shear layer tend to lie parallel to the melt flow direction (more than 95 percent in 2 direction) in contrast to the core layer in which the tend to turn to normal direction to the melt flow (about 70 percent in 2 direction).

Studying the model plate 2 is also leads to the fact that the orientation of the glass fibres mainly results from the complex melt flow during filling of the mould cavity. In addition to different local fibre orientation over the part fibre orientation also varies in the wall thickness direction.

Figures 4.23 and 4.24 illustrate the fibres orientation in 1 and 2 directions for the model plate 2, respectively.
Figure 4.21. Fibre orientation in 1 direction of the model 1 (Section view AA).

Figure 4.22. Fibre orientation in 2 direction of the model 1 (Section view AA).
Figure 4.23. Fibre orientation in 1 direction of the model 2 (Section view AA).

Figure 4.24. Fibre orientation in 2 direction of the model 2 (Section view AA).
4.2.4.3 Result Extraction (SIGMAlink)

Before importing the results from SIGMASOFT, an ABAQUS input file is required for each model that must consist of the geometry of the meshed part.

Using SIGMAlink interface to translate data from SIGMASOFT to ABAQUS and store this information based on ABAQUS input file.

In order to import data from SIGMASOFT, the ABAQUS input file was prepared just for the plate, so just the data related to the plate must be transferred with the same coordinate system in SIGMASOFT and ABAQUS.

What is required from SIGMASOFT is the fibre orientation for the completely filled stages, so all components of the orientation tensor were imported. For translating data from SIGMASOFT elements to ABAQUS, there are two methods in SIGMAlink. The first one is using the element center that means only one result is written out for each element. The second method is aiding the Gauss integration points, which is more precise. For example, in hexahedral elements (our case) there are eight Gauss integration points, therefore eight different results for each element are written out, so that there is given a quite accurate orientation profile over the element. On the other hand for hexahedral elements eight times more RAM is required to store the orientation values in contrast to only one result for each element.

4.2.5 JAVA Interface

After translating data into the ABAQUS plate geometry format, the next interface is a JAVA execution program “ABAQUSmeetsSIGMA” that was written in the IKV. The first part of the program is based on Halpin-Tsai equation (Chapter 3.2.2) for calculating the stiffness tensor for the plate.

The input data required for the JAVA program are tabulated in Table 4.3. The fibre volume fraction \( \Phi_f \) is calculated by Eq. 3.5.
Table 4.3. Data required for Halpin-Tsai equations.

<table>
<thead>
<tr>
<th>Property</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>$E_{\text{glass fiber}}$</td>
<td>72000 [Mpa]</td>
</tr>
<tr>
<td>$E_{\text{polypropylene}}$</td>
<td>1500 [Mpa]</td>
</tr>
<tr>
<td>$\nu_{\text{glass fiber}}$</td>
<td>0.2</td>
</tr>
<tr>
<td>$\nu_{\text{polypropylene}}$</td>
<td>0.35</td>
</tr>
<tr>
<td>$\Phi_f$</td>
<td>0.129</td>
</tr>
</tbody>
</table>

So, at first the JAVA program calculates the stiffness parameters of a unidirectional fibre reinforced composite by using the micromechanical Halpin-Tsai equations (Table 3.2) and store them in a text file. Then using Advani-Tucker orientation averaging method [11] the parameters in the stiffness matrix of unidirectional fibre oriented composite are adjusted to the degrees of fibre orientation for each Gauss integration point (for each element).

This way the data required for the ABAQUS user material subroutine is prepared, and can be used in the material definition part of preprocessing. This data consist of the direction of the local coordinate system and the stiffness coefficient that matches to the local coordinate system for each Gauss integration point.

4.2.6 Acoustic Simulation in ABAQUS

As mentioned earlier, the acoustic simulation of the model plates can be performed in ABAQUS. The simulation was done for three models: an unreinforced polypropylene plate, model plate 1 and model plate 2.

4.2.6.1 Preprocessing

In this stage the models of the physical problems were defined and ABAQUS input files (Appendix B and Appendix C) were created.
• **Part Geometry**

The model plate is created in SOLIDWORKS and imported as a “stl” file to ABAQUS (similar to the job done in Chapter 4.1.1). As it is shown in Figure 4.1, it is a plate in feature of 112.5 mm length, 110.5 mm width and 2 mm thickness in which two corners of the model plate are filleted by a radius of 12 mm.

• **Material Modeling**

The user subroutine was written for ABAQUS in FORTRAN. The user subroutine is based on the local coordinate systems and needs the stiffness coefficient that matches to the local coordinate system for each Gauss integration point. This information is stored in two different text files. During the acoustic simulation in ABAQUS, the FORTRAN user subroutine is called, in order to set up the local stiffness matrix for each Gauss integration point.

Due to the fact that there was no method found for defining and applying the local damping behavior for short fibre reinforced composites (at the time of writing this thesis report), the value six is chosen and assigned as the mass proportional damping factor (Alpha). In further studies this value has proven suitable to define damping for plastic materials. The stiffness proportional damping is not applicable together with a user material subroutine definition in ABAQUS, so the Beta value has to be kept zero.

• **Enmeshment and Element type**

The enmeshment and element type chosen are the same as those used in the previous simulations (Chapter 4.1.1).

• **Step involved in the modeling**

Two steps are assigned for this task:

1-Natural frequency extraction (by Lanczos eigensolver in the frequency interval of 1~2000 Hz)

2-Steady-state dynamic response analysis (in the frequency interval of 1~2000 Hz)
• Load and boundary conditions

The load and boundary conditions applied are similar to those used in Chapter 4.1.1.

4.2.6.2 Postprocessing and Discussion

In the first part of this section natural frequencies and associated modes of vibration are discussed, which give information about the deformation behavior of the system corresponding to the eigenmodes. While in the analysis of the steady state dynamics response, the effect of the harmonic force on the displacement and velocity responses is investigated.

• Analysis done in step-1 (Frequency extraction)

The first four mode shapes of the natural frequency extraction step with corresponding undamped natural frequencies are shown in Figures 4.25, 4.26 and 4.27 for the PP plate, PP GF 30 model plate 1 and 2, respectively. And also the first ten undamped natural frequencies are tabulated in Table 4.4 for these three plates.

Table 4.4. Natural resonance frequencies (Hz).

<table>
<thead>
<tr>
<th>Mode No.</th>
<th>PP</th>
<th>PP GF 30 Model1</th>
<th>PP GF 30 Model2</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>48.642</td>
<td>70.029</td>
<td>54.684</td>
</tr>
<tr>
<td>2</td>
<td>88.742</td>
<td>117.16</td>
<td>104.88</td>
</tr>
<tr>
<td>3</td>
<td>300.49</td>
<td>380.40</td>
<td>340</td>
</tr>
<tr>
<td>4</td>
<td>341.78</td>
<td>447.47</td>
<td>425.83</td>
</tr>
<tr>
<td>5</td>
<td>370.34</td>
<td>511.13</td>
<td>477.64</td>
</tr>
<tr>
<td>6</td>
<td>597.05</td>
<td>754.93</td>
<td>750.74</td>
</tr>
<tr>
<td>7</td>
<td>843.20</td>
<td>958.25</td>
<td>955.31</td>
</tr>
<tr>
<td>8</td>
<td>852.03</td>
<td>1220.8</td>
<td>1023</td>
</tr>
<tr>
<td>9</td>
<td>915.28</td>
<td>1229.3</td>
<td>1235.9</td>
</tr>
<tr>
<td>10</td>
<td>1076.3</td>
<td>1314.2</td>
<td>1319.1</td>
</tr>
<tr>
<td>…</td>
<td>…</td>
<td>…</td>
<td>…</td>
</tr>
</tbody>
</table>

As it is obvious in Table 4.4, the undamped natural frequencies are shifted to higher values from the PP plate to the model plate 2 and the model
plate 1. For example the mode shape for the PP plate is laid on 48.642 Hz in contrast to 54.684 Hz for the model plate 2 and 70.029 Hz for the model plate 1. It declares the fact that for the first seven resonances the model plate 1 behaves stiffer than the two others. The effect of reinforcement with glass fibres on the stiffness tensor of the model plate 1 and 2 is clear.

On the other hand, the contour plots of the deformed plate show how the plate is deformed relatively. Figure 4.25 shows that opposite regions are more deformed than the regions near by the fixed end. And also there are other regions with zero deformation depending on corresponding eigenform. The first and second mode shapes are similar in some ways but the differences are started from the third mode shape where the third mode shape for the PP plate is totally differed from the model plate 1 (Figure 4.26) and also this differences started for the model plate 2 from the fourth mode shape (Figure 4.27).

![Figure 4.25. Deformation of the PP model plate at the undamped resonance frequencies.](image-url)
Figure 4.26. Deformation of the PP GF 30 model plate 1 at the undamped resonance frequencies.

Figure 4.27. Deformation of the PP GF 30 model plate 2 at the undamped resonance frequencies.
• Analysis done in step-2 (Steady state dynamics)

In this section the steady state dynamics response of the model plates including material damping is achieved and the displacement, velocity and acceleration responses obtained at the node of the applying of the harmonic force are extracted.

The first four deformations of the PP plate, PP GF 30 model plate 1 and 2 at peak values of damped resonance frequencies are shown in Figure 4.28, 4.29 and 4.30 for the first four damped resonance frequencies.

Figure 4.28. Deformation of the PP model plate at the peak values of the damped resonance frequencies.
Figure 4.29. Deformation of the PP GF 30 model plate at the peak values of the damped resonance frequencies.

Figure 4.30. Deformation of the PP GF 30 model plate at the peak values of the damped resonance frequencies.
Figure 4.31, 4.32 and 4.33 shows the displacement, velocity and acceleration responses of the PP plate, PP GF 30 model plate 1 and 2 to the harmonic force, respectively.

**Figure 4.31.** Displacement, velocity and acceleration responses vs. frequency for the PP model plate.

**Figure 4.32.** Displacement, velocity and acceleration responses vs. frequency for the PP GF 30 model plate 1.
In Figure 4.34, the effect of short glass fibres in reinforcing the plastic is clearer precisely, where in the velocity response of the three models to the harmonic force is compared. The first resonance frequency is shifted from 49 Hz for the PP plate to higher values for the model plate 2 and 1 with 55 Hz and 70 Hz, respectively.

For a better comparison between the model plate 1 and 2 responses, Figure 4.35 shows the velocity responses of the model plate 1 and 2 in the linear scaling for the x axes. In this Figure, two resonance peaks are focused; at the first resonance peak the model plate 2 is pursuing the model plate 1 with 15Hz frequency lag in contrast to the sixth resonance peak in which the velocity peak in the model 1 is situated at 755 Hz while the same peak response for the model 2 is laid on 790 Hz. These differences come from the mode shape and the corresponding element of stiffness tensor of the composite model that some times the model plate 1 shows stiffer than the other and sometimes vice versa. So the fibre orientations play an important role for the harmonic response of parts.

Figures 4.36 and 4.37 compare the deformation of the first and sixth response peak, respectively for the model plate 1 and 2.
4.34. Comparison between the PP model plate, PP GF 30 model plate 1 and 2 velocity responses.

4.35. Comparison between the PP GF 30 model plate 1 and 2 velocity responses.
Figure 4.36. Comparison of the deformation at the first resonance peak for the model plate 1 and 2.

Figure 4.37. Comparison of the deformation at the sixth resonance peak for model plate 1 and 2.
5 Conclusion and Further Works

The effect of anisotropy on the simulations of this thesis work shows that it is possible to regard the local stiffness affected by the fibre orientation. However due to lower damping behavior in the fibre orientation and higher damping behavior in the transverse fibre direction, there is no precise method for describing this damping behavior of short glass fibre reinforced thermoplastics, at the time of writing this report.

Finite element method is a useful numerical method for modeling the composites and also ABAQUS is suitable for modeling the steady state dynamic analysis and extracts probably a more accurate result if the injection moulding process is taken to account. This can be done by simulation of the fibre orientations in a process simulation tool like SIGMASOFT and exporting the resulting stiffness data to an ABAQUS user material subroutine using micromechanical models.

Further work could take into consideration the following points;

- Some experimental analysis can be done on the real specimens that are available at IKV, in order to check and verify the results from the simulations of this thesis.

- Viscoelastic theory should be more refined to consider the frequency dependent composite material behavior for stiffness and damping.

- A material model should be established that makes possible the anisotropic simulation in ABAQUS.
6 References


10. Schmachtenberg E. Brandt M., (2005), Mechnische Auslegung von kurzfaserverstärkten Spritzgussbauteilen vollständig in 3D.


A. ABAQUS input file for the polyamide model plate

**Input file for Acoustic Simulation of a polyamide Model Plate**

**Heading**

**Job name: polyamideAcoustic Sim: polyamidePlate**

**PART Module **********************************************

**Part, name=PA Plate**

*Node

1, 56., 9.42900467, 0.
2, 56., 46., 0.
3, 5.1550765, 11.1808453, 0.
.
.
.
3578, -44.5250053, 51.9863625, 2.
3579, -48.3500023, 51.9909096, 2.
3580, -52.1749992, 51.9940681, 2.

*Element, type=C3D8

1, 31, 154, 11, 1, 926, 1049, 906, 896
2, 30, 155, 154, 31, 925, 1050, 926
3, 29, 156, 155, 30, 924, 1051, 1050, 925
.
.
.
2512, 1930, 1929, 2684, 2683, 2825, 2824, 3579, 3578
2513, 1929, 1928, 2685, 2684, 2824, 2823, 3580, 3579
2514, 1928, 1800, 1927, 2685, 2823, 2695, 2822, 3580

*Nset, nset=_PeakedSet9, internal, generate

1, 3580, 1

*Elset, elset=_PeakedSet9, internal, generate
** Region: (Section-1: Peaked) **
* Elset, elset=_PeakedSet9, internal, generate
  1, 2514, 1
** Section: Section-1 **
* Solid Section, elset=_PeakedSet9, material=Plastic
  1.
* End Part
**

** ASSEMBLY Module **************************************************************************
**
* Assembly, name=Assembly
**
* Instance, name=PA Plate, part=PA Plate
* End Instance
**

* Nset, nset=_PeakedSet4, internal, instance=Plate-1
  2691,
* Nset, nset=_PeakedSet5, internal, instance=Plate-1
  9, 10, 111, 112, 113, 114, 115, 116, 117, 118, 119, 120, 121, 122, 123, 124
  .
  .
  2803, 2804, 2805, 2806, 2807, 2808, 2809, 2810, 2811, 2812, 2813, 2814, 2815, 2816, 2817, 2818
  2819, 2820, 2821, 2822
* Elset, elset=_PeakedSet5, internal, instance=Plate-1
  406, 422, 438, 454, 470, 486, 502, 518, 534, 550, 566, 582, 598, 614, 630, 646
  .
  .
  2210, 2226, 2242, 2258, 2274, 2290, 2306, 2322, 2338, 2354, 2370, 2386, 2402, 2418, 2434, 2450
Material, name=PA Plastic

Density
1.144e-09.

Elastic, moduli=LONG TERM
380., 0.28

Viscoelastic, frequency=TABULAR

<table>
<thead>
<tr>
<th>Frequency</th>
<th>Real</th>
<th>Imaginary</th>
<th>Frequency</th>
<th>Real</th>
<th>Imaginary</th>
<th>Frequency</th>
<th>Real</th>
<th>Imaginary</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.0490263</td>
<td>0.0609153</td>
<td>0.0487705</td>
<td>0.0557973</td>
<td>0.001</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>0.0838516</td>
<td>0.409142</td>
<td>0.0509866</td>
<td>0.0549427</td>
<td>10.011</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>0.0874009</td>
<td>0.444305</td>
<td>0.0743564</td>
<td>0.0806033</td>
<td>20.021</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Frequency</th>
<th>Real</th>
<th>Imaginary</th>
<th>Frequency</th>
<th>Real</th>
<th>Imaginary</th>
<th>Frequency</th>
<th>Real</th>
<th>Imaginary</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.235659</td>
<td>-1.05676</td>
<td>0.0844816</td>
<td>-0.057756</td>
<td>9979.98</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>0.23572</td>
<td>-1.0572</td>
<td>0.0845022</td>
<td>-0.0573199</td>
<td>9989.99</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>0.23578</td>
<td>-1.05765</td>
<td>0.0845228</td>
<td>-0.0568842</td>
<td>10000.</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

** Step Module *****************************************************
** STEP: FIND THE NATURAL FREQUENCY
**
*Step, name="FIND THE NATURAL FREQUENCY", perturbation FIND THE NATURAL FREQUENCY
*Frequency, eigensolver=Lanczos, acoustic coupling=on, normalization=displacement, number interval=1, bias=1., 1., 10000., , ,
**
** Load Module *****************************************************
** BOUNDARY CONDITIONS  ******************************************************************
**
** Name: CLAMPED Type: Symmetry/Antisymmetry/Encastre
*Boundary
  _PeakedSet6, ENCASTRE
**
** OUTPUT REQUESTS
**
*Restart, write, frequency=0
**
** FIELD OUTPUT: F-Output-1
**
*Output, field
*Node Output
A, U, V
**
*End Step
**  ******************************************************************
**
** STEP: APPLYING STEADY STATE DYNAMICS
**
*Step, name="APPLYING STEADY STATE DYNAMICS", perturbation
*Steady State Dynamics, direct, frequency scale=LINEAR
  1., 10000., 10000, 1.
**
** Load Module  ******************************************************************
**
** BOUNDARY CONDITIONS
**
** Name: CLAMPED Type: Symmetry/Antisymmetry/Encastre
*Boundary
  _PeakedSet5, ENCASTRE
**
** LOADS
**
** Name: CLOAD  Type: Concentrated force
* Cload, load case=1
  _ PeakedSet 4, 3, 1.
**
** OUTPUT REQUESTS
**
**
** FIELD OUTPUT: F-Output-2
**
* Output, field
* Node Output
A, U, V
**
** HISTORY OUTPUT: H-Output
**
* Output, history
Model output
GA, GV
* End Step
B. ABAQUS input file for the PP GF 30 model plate 1

**Input file for Acoustic Simulation of the PP-GF-30-ModelPlate1

*Heading

** Job name: PP-GF-30-Acoustic-Sim-model1: PP-GF-30-Plate

**

** PART Module *************************************************************

**

*Part, name= PP-GF-30-Plate
*End Part

**

**

** ASSEMBLY Module *************************************************************

**

*Assembly, name=Assembly

**

*Instance, name= PP-GF-30-Plate-1, part= PP-GF-30-Plate

*Node

1, 112.5, 40.102066, 2.

2, 66.465416, 43.9353981, 2.

3, 67.2839279, 0., 2.

., ., ., .,

., ., ., .,

., ., ., .,

3462, 53.3212395, 102.945633, 0.

3463, 57.3190689, 102.886826, 0.

3464, 61.3423729, 102.828407, 0.

*Element, type=C3D8

1, 52, 149, 11, 1, 918, 1015, 877, 867

2, 149, 150, 12, 11, 1015, 1016, 878, 877

3, 150, 151, 13, 12, 1016, 1017, 879, 878
**Nset, nset=_PeakedSet2, internal, generate
1, 3464, 1
*Elset, elset=_PeakedSet2, internal, generate
1, 2430, 1
** Region: (Section-1:Peaked)
*Elset, elset=_PeakedSet2, internal, generate
1, 2430, 1
**

*ORIENTATION, SYSTEM=USER, NAME=ORI
**
** Section: Section-1
*Solid Section, elset=_PeakedSet2, material=PP_30_GF_Model1, ORIENTATION=ORI
1.,
*End Instance
**

*Nset, nset=_PeakedSet19, internal, instance=ModelPlate1_Assem-1
3, 4, 9, 32, 33, 34, 35, 36, 37, 38, 39, 40, 41, 42, 109, 110
.
.
.
2633, 2634, 2635, 2636, 2637, 2638, 2639, 2640, 2707, 2708, 2709, 2710, 2711, 2712, 2713, 2714
2715, 2716, 2717, 2718, 2719, 2720, 2721, 2722
*Elset, elset=_PeakedSet19, internal, instance=ModelPlate1_Assem-1
121, 122, 123, 124, 125, 126, 127, 128, 129, 130, 131, 132, 352, 379, 380, 381

81
** Property Module ************************************************************
**
** MATERIALS
**
*Material, name=PP_30_GF_Model1
*Damping, alpha=6.
*DENSITY
1.14E-9
*USER MATERIAL, CONSTANTS=0
**
*DEPVAR
10
**
** Step Module *******************************************************
**
** STEP: FIND THE NATURAL FREQUENCY
**
*Step, name="FIND THE NATURAL FREQUENCY", perturbation FIND THE NATURAL FREQUENCY
*Frequency, eigensolver=Lanczos, acoustic coupling=on, normalization=displacement, number interval=1, bias=1., 1., 2000., , ,
**
** Load Module ************************************************************
**
** BOUNDARY CONDITIONS *****************************************************
**
** Name: CLAMPED Type: Symmetry/Antisymmetry/Encastre
*Boundary
  _PeakedSet6, ENCASTRE
**
** OUTPUT REQUESTS
**
*Restart, write, frequency=0
**
** FIELD OUTPUT: F-Output-1
**
*Output, field
*Node Output
A, U, V
**
*End Step
**  
**  
** STEP: APPLYING STEADY STATE DYNAMICS
**
*Step, name="APPLYING STEADY STATE DYNAMICS", perturbation
*Steady State Dynamics, direct, frequency scale=LINEAR
**
** Load Module *********************************************
**
** BOUNDARY CONDITIONS
**
** Name: CLAMPED Type: Symmetry/Antisymmetry/Encastre
*Boundary
  _PeakedSet5, ENCASTRE
**
** LOADS
**
** Name: CLOAD Type: Concentrated force
*Cload, load case=1
_PeakedSet 4, 3, 1.
**
** OUTPUT REQUESTS
**
**
** FIELD OUTPUT: F-Output-2
**
*Output, field
*Node Output
A, U, V
**
** HISTORY OUTPUT: H-Output
**
*Output, history
Model output
GA, GV
*End Step
C. ABAQUS input file for the PP GF 30 model plate 2

**Input file for Acoustic Simulation of the PP-GF-30-ModelPlate2

*Heading
** Job name: PP-GF-30-Acoustic-Sim-model2: PP-GF-30-Plate
**

** PART Module ****************************************************
**
*Part, name= PP-GF-30-Plate
*End Part
**

**

** ASSEMBLY Module *****************************************************
**
*Assembly, name=Assembly
**
*Instance, name = PP-GF-30-Plate-1, part = PP-GF-30-Plate
*Node

1, 112.5, 40.102066, 2.
2, 66.465416, 43.9353981, 2.
3, 67.2839279, 0., 2.
., ., ., .,
., ., ., .,
., ., ., .,
3462, 53.3212395, 102.945633, 0.
3463, 57.3190689, 102.886826, 0.
3464, 61.3423729, 102.828407, 0.
*Element, type=C3D8

1, 52, 149, 11, 1, 918, 1015, 877, 867
2, 149, 150, 12, 11, 1015, 1016, 878, 877
3, 150, 151, 13, 12, 1016, 1017, 879, 878
*Nset, nset=_PeakedSet2, internal, generate
  1, 3464, 1
*Elset, elset=_PeakedSet2, internal, generate
  1, 2430, 1
** Region: (Section-1: Peaked)
*Elset, elset=_PeakedSet2, internal, generate
  1, 2430, 1
**
*ORIENTATION, SYSTEM=USER, NAME=ORI
**
** Section: Section-1
*Solid Section, elset=_PeakedSet2, material=PP_30_GF_Model2, ORIENTATION=ORI
  1.
*End Instance
**
*Nset, nset=_PeakedSet19, internal, instance=ModelPlate1_Assem-1
  3, 4, 9, 32, 33, 34, 35, 36, 37, 38, 39, 40, 41, 42, 109, 110
  .
  .
  2633, 2634, 2635, 2636, 2637, 2638, 2639, 2640, 2707, 2708,
  2709, 2710, 2711, 2712, 2713, 2714
  2715, 2716, 2717, 2718, 2719, 2720, 2721, 2722
*Elset, elset=_PeakedSet19, internal, instance=ModelPlate1_Assem-1
  121, 122, 123, 124, 125, 126, 127, 128, 129, 130,
  131, 132, 352, 379, 380, 381
*End Assembly*

** Property Module ******************************************************
**
** MATERIALS
**
*Material, name=PP_30_GF_Model2
*Damping, alpha=6.
*DENSITY
1.14E-9
*USER MATERIAL, CONSTANTS=0
**
*DEPVAR
10
**
** Step Module *********************************************************
**
** STEP: FIND THE NATURAL FREQUENCY
**
*Step, name="FIND THE NATURAL FREQUENCY", perturbation
FIND THE NATURAL FREQUENCY
*Frequency, eigensolver=Lanczos, acoustic coupling=on,
normalization=displacement, number interval=1, bias=1.
, 1., 2000., , ,
**
** Load Module *********************************************************
**
** BOUNDARY CONDITIONS ***************************************************
**
** Name: CLAMPED Type: Symmetry/Antisymmetry/Encastre
*Boundary
  PeakedSet 6, ENCASTRE
**
** OUTPUT REQUESTS
**
*Restart, write, frequency=0
**
** FIELD OUTPUT: F-Output-1
**
*Output, field
*Node Output
A, U, V
**
*End Step
**  ---------------------------------------------------------
**
** STEP: APPLYING STEADY STATE DYNAMICS
**
*Step, name="APPLYING STEADY STATE DYNAMICS", perturbation
*Steady State Dynamics, direct, frequency scale=LINEAR
**
** Load Module ****************************************************
**
** BOUNDARY CONDITIONS
**
** Name: CLAMPED Type: Symmetry/Antisymmetry/Encastre
*Boundary
  PeakedSet 5, ENCASTRE
**
** LOADS
**
** Name: CLOAD Type: Concentrated force
*Cload, load case=1
_PeakedSet 4, 3, 1.
**
** OUTPUT REQUESTS
**
**
** FIELD OUTPUT: F-Output-2
**
*Output, field
*Node Output
A, U, V
**
** HISTORY OUTPUT: H-Output
**
*Output, history
Model output
GA, GV
*End Step
D. Material Data in SIGMASOFT for the PP GF 30 model plates

<table>
<thead>
<tr>
<th>Material type</th>
<th>plastic</th>
</tr>
</thead>
<tbody>
<tr>
<td>Initial Temperature</td>
<td>240 °C</td>
</tr>
<tr>
<td>Lambda</td>
<td>0.27 W/mK</td>
</tr>
<tr>
<td>$C_p$</td>
<td>1390 J/kgK</td>
</tr>
<tr>
<td>No-Flow-Temperature</td>
<td>130 °C</td>
</tr>
</tbody>
</table>

**Rheology (Carreau WLF) parameters**

| Zero Shear Rate (dyn.) Visc. P1 | 8.92 e+06 Pa s |
| Reciprocal Trans. Shear Rate P2 | 5202.40 s |
| Exponent P3 | 0.74 |
| Reference Temperature T0 | 240 °C |
| Standard Temperature Ts | 96.72 °C |

**pvT Schmidt Coefficients**

**Low Temperature Region**

- PF1 59525.8 bar cm$^3$/g
- PF2 0.49 bar cm$^3$/g K
- PF3 3142.82 bar
- PF4 67465.30 bar

**Transition Region**

- PF5 2.32e-07 cm$^3$/g
- PF6 0.098 1/K
- PF7 2.5719e-03 1/bar

**High Temperature Region**

- PS1 46632.30 bar cm$^3$/g
- PS2 0.70 bar cm$^3$/g K
- PS3 1258.03 bar
- PS4 51093.70 bar

**Limit Transition**

- PK1 134 °C
- PK2 0.0223 °C/bar

**Fibre General Properties**

- Aspect Ratio | 25 |
- Geometry Parameter | 0.997 |
- Weight Fraction | 15.00 % |
- Interaction Coefficient | 1.00e-02 |