Abstract
The performance of the Elite® advanced resonant power toothbrush is measured in terms of oral biofilm removal and improvements in oral health over time. These are strongly related to the mechanical performance of the brush head and bristles located on it. To find an optimum resonant structure, which provides both the required performance and a specified clinical experience, the transient behavior of the toothbrush is modeled.

To optimize the resonant structure, we are using an APDL built FEM model. Using this model we are able to perform Modal, Harmonic or Transient analysis within ANSYS Mechanical (Release 7.1). To save time and expense, the dynamics of the brush head are calculated by using a modal analysis in FEM, and exported as eigen vectors to Matlab. In Matlab this data is translated to a state-space formulation and used as a transfer function of a linear system in Matlab Simulink. This allows us to apply various excitations and look at time domain responses of our power toothbrush.

Introduction
To evaluate the toothbrush response in time domain, one needs to do a transient dynamic analysis. ANSYS offers three methods of transient dynamic analysis: Full transient, Modal superposition method, and Reduced method. These methods differ in calculation method, and also in range of applicable types of load. In this paper we compare our results from ANSYS with transient calculations done in Matlab Simulink.

When using ANSYS for transient analysis, we have faced the following issues, which make our analysis more complex:

1. Input signal definition:
   Tabular input requires pre-data processing in excel versus a direct loading from the measurement. For simple waveforms the ANSYS tabular format was sufficient.

2. Time of calculation:
   There is significant difference in calculation time between ANSYS transient analysis methods (minutes versus hours). Using modal decomposition with the Matlab Simulink transient analysis is in the range of seconds.

3. Reviewing results:
   ANSYS requires more steps to transfer information from analysis into a report.

To decrease calculation time and simplify user interaction with the model, one can make use of modal decomposition, extracting the linear representation of the dynamic system in state space form into Matlab.
**FEM model development**

The Power toothbrush model, used for presented results, is built for dynamic analysis, and therefore the dynamic structure is represented as a simplified lumped mass model describing the flexibilities and mass distribution of the brush head. The model was done in a parametric way to be able to perform sensitivity study on a variety of design parameters.

APDL language from ANSYS is a very powerful tool, and allows the user to make a variety of sensitivity studies in a short time.

![Figure 1. Element plot of Elite® FEM model](image1.png)  ![Figure 2. Brush rotational mode](image2.png)

**Figure 1. Element plot of Elite® FEM model**  **Figure 2. Brush rotational mode**

Figure 1 illustrates different elements used in the model:
- purple represents stiff shell of bristles (SHELL63)
- green color represents beams of the plastic brush stem (BEAM4)
- blue represents beams of the steel shaft (BEAM4)
- red represents steel springs (SHELL63)

Then some point masses are defined to be able to shift the center of mass of selected parts or insert the excitation force at different locations. The model is suitable for modal analysis, but it would need refinement to have a reliable stress analysis.

Modal analysis gives the user the sensitivities of resonant frequencies to design parameters. Harmonic analysis is used when one is interested in brush head amplitude due to defined excitation magnitude, position, and direction. If one wants to have comparison of displacements of one point for different design parameters, he needs to define several key points in the model. These will remain the same for every calculation, and for each calculation the model gets the number of node at this reference key point, and uses it to get the right results. Figure 3 depicts reference points used for the structure output – displacements KBL (344), KBC (255), KBR (377). Figure 4 shows the reference point used for positioning the excitation force F (7).
To do a transient analysis of dynamic structure we use reference points shown in Figures 3 and 4 to extract eigen frequencies and eigen vectors from results of modal analysis. We do this for specified number of modes in all six degrees of freedom. Figure 5 illustrates the summation of the contribution of all extracted modes in total frequency response function. Figure 6 depicts the separate contribution of each mode to the final response function.
Transient analysis using ANSYS

ANSYS offers three methods of transient dynamic analysis: Full transient, modal superposition, and reduced methods. To investigate the performance of power toothbrushes, one needs to define the custom load profile of non-harmonic excitation force.

Figure 7. Segment of custom defined excitation force

This is done in the presented model by tabular input. The Full transient analysis allows the use of tabular input directly, and when using modal superposition, one needs to use loops and assign other parameters to use the tabular input (see code samples in Table 1, Table 2). Both methods, with the correct input, give identical results (see Figure 13). How to define the correct data, such as the integration method, is critical to getting the correct result. That is not addressed in this paper.
**Full transient analysis**

The full method uses the full system matrices to calculate the transient response (no matrix reduction). It is the most general of the three methods because it allows all types of nonlinearities to be included. If you do not want to include any nonlinearities, you should consider using one of the other methods because the full method is also the most expensive method of the three.

The advantages of the full method are:

- It allows all types of nonlinearities.
- It is easy to use, because you do not have to worry about choosing master degrees of freedom or mode shapes.
- It uses full matrices, so no mass matrix approximation is involved.
- All displacements and stresses are calculated in a single pass.
- It accepts all types of loads: nodal forces, imposed (nonzero) displacements (although not recommended), and element loads (pressures and temperatures) and allows tabular boundary condition specification via TABLE type array parameters.
- It allows effective use of solid-model loads.

The main disadvantage of the full method is that it is more expensive than either of the other methods.

### Table 1: Example of full transient analysis with tabular force input

<table>
<thead>
<tr>
<th>Line</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>*IF, transient, EQ, 1, THEN</td>
<td>maximum number of results sets</td>
</tr>
<tr>
<td>/CONFIG, NRES, 500000</td>
<td></td>
</tr>
<tr>
<td>/SOLU</td>
<td></td>
</tr>
<tr>
<td>ANTYPE, TRANS</td>
<td></td>
</tr>
<tr>
<td>TRNOPT, FULL</td>
<td></td>
</tr>
<tr>
<td>*DIM, force, table, 5001, 1, 1, TIME,,</td>
<td>defining the input table dependent on time</td>
</tr>
<tr>
<td>*TREAD, force, input4, txt,,</td>
<td>reading input values</td>
</tr>
<tr>
<td>OUTRES, NSOL, ALL,</td>
<td></td>
</tr>
<tr>
<td>DMPRAT, 0.01</td>
<td></td>
</tr>
<tr>
<td>TM_START=0.0</td>
<td>Start time</td>
</tr>
<tr>
<td>TM_END=0.1</td>
<td>End time</td>
</tr>
<tr>
<td>TM_INCR=1E-4</td>
<td>Step size</td>
</tr>
<tr>
<td>*DO, TM, TM_START, TM_END, TM_INCR</td>
<td>Assigning the force value from table to FZ dependent on time</td>
</tr>
<tr>
<td>TIME, TM</td>
<td></td>
</tr>
<tr>
<td>F_N_F.FZ,%force%,,</td>
<td></td>
</tr>
<tr>
<td>SOLVE</td>
<td>Solve each step</td>
</tr>
<tr>
<td>*ENDDO</td>
<td></td>
</tr>
<tr>
<td>FINISH</td>
<td></td>
</tr>
</tbody>
</table>
Modal superposition analysis

The mode superposition method sums factored mode shapes (eigenvectors) from a modal analysis to calculate the structure's response. This is the only method available in the ANSYS Professional program.

Its advantages are:

- It is faster and less expensive than the reduced or the full method for many problems.
- Element loads applied in the preceding modal analysis can be applied in the transient dynamic analysis.
- It accepts modal damping.

The disadvantages of the mode superposition method are:

- The time step must remain constant throughout the transient, so automatic time stepping is not allowed.
- The only nonlinearity allowed is simple node-to-node contact.
- It does not accept imposed (nonzero) displacements.

Table 2: Example of modal superposition transient analysis with tabular force input

<table>
<thead>
<tr>
<th>IF transient, EQ, 1, THEN</th>
</tr>
</thead>
<tbody>
<tr>
<td>ANTYPE, TRANS</td>
</tr>
<tr>
<td>TRNOPT, MSUP, 10, DAMP</td>
</tr>
<tr>
<td>TIMINT, ON, STRUCT</td>
</tr>
<tr>
<td>*DIM, force, table, 5001, 1, 1, TIME,,</td>
</tr>
<tr>
<td>*TREAD, force, input4, txt,,</td>
</tr>
<tr>
<td>DELTIM, 1E-4, 1E-6, 1E-4, ON</td>
</tr>
<tr>
<td>AUTOTS, OFF</td>
</tr>
<tr>
<td>OUTRES, OFF</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>OUTRES, 1</th>
</tr>
</thead>
<tbody>
<tr>
<td>TM_START=1E-4</td>
</tr>
<tr>
<td>TM_END=0.0</td>
</tr>
<tr>
<td>TM_INCR=1E-4</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>i=0</th>
</tr>
</thead>
<tbody>
<tr>
<td>*DO, TM, TM_START, TM_END, TM_INCR</td>
</tr>
<tr>
<td>i=i+1</td>
</tr>
<tr>
<td>OUTRES, 1</td>
</tr>
<tr>
<td>NSUBST, 1</td>
</tr>
<tr>
<td>TIME, TM</td>
</tr>
<tr>
<td>value=force(TM)</td>
</tr>
<tr>
<td>*STATUS, VALUE, 1,, 1,, 1,, 1,,</td>
</tr>
<tr>
<td>F, N, F, FZ, value,,</td>
</tr>
<tr>
<td>SOLVE</td>
</tr>
<tr>
<td>*ENDDO</td>
</tr>
<tr>
<td>FINISH</td>
</tr>
</tbody>
</table>

Number of modes included in modal superposition defining the input table (timing from program)

Limit number of substeps timing reading input values
Matlab Simulink model

It is assumed that the system is described by

\[ M\ddot{q}(t) + B\dot{q}(t) + Kq(t) = f(t) \]  \[1\]

with \( q \) being the \( [n] \) vector of displacements of physical DOFs, \( f \) the \( [n] \) vector of forces, \( M \) and \( K \) the positive definite mass and stiffness matrix and \( B \) the symmetric damping matrix. The dynamics of a system is often described in a state space form because this is the efficient way of representing the dynamics.

\[ \dot{x} = Ax + Bf \]  \[2\]
\[ q = Cx + Df \]

where \( x \) – states of the system, \( A \) – system matrix, \( B \) – input matrix, \( f \) – input (in our case force), \( q \) – output, \( C \) – output matrix, \( D \) – identity matrix (in our case is assumed to be zero). Rewriting equation 1 of the system in state space using modal information (eigenvalues and eigenvectors),

\[
\begin{bmatrix}
0 \\
\vdots \\
0 \\
\end{bmatrix}
\begin{bmatrix}
I \\
\vdots \\
I \\
\end{bmatrix}
\begin{bmatrix}
k_1 & \cdots & d_1 & \cdots & 0 \\
m_1 & \cdots & m_1 & \cdots & 0 \\
\vdots & \ddots & \vdots & \ddots & \vdots \\
0 & \cdots & k_n & \cdots & 0 \\
m_n & \cdots & m_n & \cdots & 0 \\
\end{bmatrix}
\begin{bmatrix}
x \\
\end{bmatrix}
\begin{bmatrix}
0 \\
0 \\
\end{bmatrix}
\begin{bmatrix}
f \\
U^T \\
\end{bmatrix}
\]  \[3\]

\[
\begin{bmatrix}
q \\
q \\
\end{bmatrix}
= \begin{bmatrix}
U & 0 \\
0 & U \end{bmatrix}
\begin{bmatrix}
0 \\
0 \\
\end{bmatrix}
\begin{bmatrix}
x \\
f \\
\end{bmatrix}
\]  \[3\]

Or using \( \omega_i^2 = \frac{k_i}{m_i} \) and \( 2\xi_i\omega_i = \frac{d_i}{m_i} \) results in

\[
\begin{bmatrix}
0 \\
\vdots \\
0 \\
\end{bmatrix}
\begin{bmatrix}
I \\
\vdots \\
I \\
\end{bmatrix}
\begin{bmatrix}
-\omega_1^2 & \cdots & 0 & \cdots & -2\xi_1\omega_1 & \cdots & 0 \\
m_1 & \cdots & m_1 & \cdots & \ddots & \cdots & \vdots \\
\vdots & \ddots & \vdots & \ddots & \ddots & \ddots & \vdots \\
0 & \cdots & -\omega_n^2 & 0 & \cdots & -2\xi_n\omega_n & \cdots & 0 \\
m_n & \cdots & m_n & \cdots & \ddots & \ddots & \ddots & \vdots \\
\end{bmatrix}
\begin{bmatrix}
x \\
\end{bmatrix}
\begin{bmatrix}
0 \\
0 \\
\end{bmatrix}
\begin{bmatrix}
f \\
U^T \\
\end{bmatrix}
\]  \[4\]

\[
\begin{bmatrix}
q \\
q \\
\end{bmatrix}
= \begin{bmatrix}
U & 0 \\
0 & U \end{bmatrix}
\begin{bmatrix}
0 \\
0 \\
\end{bmatrix}
\begin{bmatrix}
x \\
f \\
\end{bmatrix}
\]  \[4\]
ξᵢ is the relative modal damping of mode i and ωᵢ is the eigenfrequency of mode i, with i=1,...,n, where n is the number of modes. Now we have used q(t) = Uz(t) with U = [u₁, u₂, ..., uₙ] meaning that for the calculation of the physical DOF the eigenvectors all of the n calculated modes are used. If we are interested in the contributions of one mode, for example mode m, than we should use U, U = [0, ..., uᵣ₉, ..., 0] or in other words, use only eigenvector m. We can see in equation 4 that the dynamics of the brush head can be described with eigenfrequencies ωᵢ, eigenvectors uᵢ, and the relative damping ξᵢ. Now we are able to construct an ABCD-system describing the dynamics of the system from force on DOF k to movements of DOF ᵐ taking the dynamics of n modes into account by just using 

\[
\begin{bmatrix}
    u₁ \\
    u₂ \\
    \vdots \\
    uₙ
\end{bmatrix}
\] and ωᵢ with i=1,...,n. These eigenvectors and eigenvalues are calculated using a modal analysis in ANSYS.

They are extracted from ANSYS as text files that contain eigenvectors and eigenvalues (and some additional information). The text files are named name.1, name.2, ..., name.10 for the first 10 mode shapes, name.evl for eigenvalues and name.dof for the DOF name.

Because a modal analysis assumes no damping, no damping data is obtained doing the FEM (modal) analysis. To approximate a small amount of material damping we apply a little modal damping of 1% relative modal damping (ξ). For larger amounts of damping the shape of the eigenvectors changes and the undamped approximation becomes less accurate. Note, modal damping ξᵢ damps the movement of a certain mode (shape). It should be clear that a local damper (for example the seal) can not be described with modal damping. Summarizing, it is difficult to translate modal damping to physical damping (and the other way around). Using little modal damping to describe the material damping is the correct approximation.

Now the eigenvalue and eigenvector data, saved in text files, together with a chosen relative modal damping (ξ) are used in the Matlab to create ABCD-system using equation 4. This ABCD-system describes the dynamics of the system from selected inputs to selected outputs making dynamic an investigation on the Elite® brush head possible. For more information on modal decomposition refer to [1,2].

![Figure 8. Transfer functions in Simulink](image)

Figure 8 shows how the ABCD representation of dynamic system was separated to single responses. For example sys(22,3) is a response of point KBR to a excitation point in magnitude of the rotational component around axis x.
Integrating the dynamics of the brush head into the total system is depicted in Figure 9. On the left we start with defining the excitation using Matlab Simulink Signal builder. This allows us to create our own signal or record the real frequency content of excitation during operation. Then we choose what DOF at what point we want to observe and run the simulation.

Figure 9. Toothbrush system model
Results comparison

To select which method, Matlab modal decomposition or ANSYS transient analysis, to use in our development, we have defined the two most important parameters:

Time expense: Time of calculation and transformation of results into report

Simulation accuracy: Fit between simulation result and laboratory measurement (not shown for confidentiality reasons)

From previous chapters one should see that we always have to calculate the linear dynamic model in ANSYS, using modal analysis. Then, if we integrate the result in Matlab, we can start adding nonlinearities, customize the excitation input, and prepare a variety of output formats. Therefore, that would be a choice when one needs to do a lot of investigation for one dynamic system. In Figure 9 we see that we had modules as “signal library” with predefined excitation waveforms, and dynamic models solved for different output (measurement) points (KBL, KBC, KBR).

Saying that ANSYS is more time-consuming for calculating our transient response is not really fair. If we calculate modal vectors, and leave them in the working directory, then the calculation time of ANSYS transient analysis would be the same as that of the Matlab method. Accuracy of results is expected to be very good, mostly because matrices are calculated with the same method as in ANSYS (Figure 13).

![Figure 10. Time domain performance – Matlab](image)

**Damping = 1 %**

![Figure 12. Time domain performance – ANSYS](image)

**(Modal superposition) Damping = 1 %**

The Full transient analysis in ANSYS gives different results in the magnitude of structure response Figure 12. This method incorporates different methods of solution for equation of motion [1]. The full solution method solves equation [1] directly and makes no additional assumptions. In contrast the mode superposition method uses the natural frequencies and mode shapes of a linear structure to predict the response to transient forcing functions. We assume that the difference in full transient calculation is due to initial conditions, and if we would let ANSYS calculate the transient longer it would settle on the same values as the modal superposition method. This was not feasible due to time restrictions, and restrictions on our hard drive space.
References

[1] A.H.Koevoets: *Brush head analysis, mode dynamics*
POHC 12NC Number: 4235 006 07241-198
Philips CFT (Centrum for manufacturing and technology)
2003-01-14 The Netherlands (CTB595-03-1074)