Dynamics – Machine-Induced Vibrations

The following RF-DYNAM Pro example presents two engines which induce vibrations on a steel frame by means of rotating unbalanced masses. Both engines work with a drive of 330 rpm. The mass of the rotor is 700 lb, and the eccentricity $e$ is 0.75 in. The mass of a single engine is 1 ton. The load from the engine and unbalanced mass are to be distributed over the adjacent members.

\[
\text{dead load of a single machine} \quad m_{m} = 1.11 \text{ ton} \quad F_{m} = m_{m} \cdot g = 2.22 \text{ kip}
\]

as member load
\[
F_{m} = \frac{F}{l} = 0.066 \frac{\text{kip}}{\text{ft}}
\]

By using the eccentricity and the rotor mass, is it possible to calculate the rotational force which acts periodically on the steel frame. This load is distributed as member loads on the adjacent members.

\[
f_{r} = \frac{330}{60} = 5.5 \text{ Hz} \quad m_R = 700 \text{ lb} \quad e = 0.75 \text{ in}
\]

\[
F_{r} = (2 \pi \cdot f_{r})^{2} \cdot e \cdot m_{R} = 1.624 \text{ kip} \quad l = 33.6 \text{ ft}
\]

induced load as member load
\[
F = \frac{F_{r}}{l} = 0.048 \frac{\text{kip}}{\text{ft}}
\]

Model
Next step for the analysis is to create the load cases for the different types of loads.

**Load Case 1: Dead Load**

LC1: dead load
Loads [kN]

**Load Case 2: Dead Load of Engines**

LC2: dead loads engines
Loads [kN/ft]
Load Case 3: Vertical Force of Induced Load

Load Case 4: Horizontal Force of Induced Load
RF-DYNAM Pro

After defining the load cases, we open the RF-DYNAM PRO module to perform the first step, which is an eigenvalue analysis. We can then compare the natural frequency of the structure with the induced frequency of the engines. If these frequencies are close together, then a resonance effect is possible.

Under the Mass Combinations tab, we create two mass cases. For Mass Case 1, we use Load Case 1. For Mass Case 2, we use Load Case 2. Then a mass combination is to be created with these two mass cases. Due to the permanent loads, we combine the mass cases with the factor of 1.00.
Now we switch to the *Natural Vibration Cases* tab. In this window it is possible to choose the number of eigenvalues for the analysis. For the calculation of the eigenvalues, we select the first mass combination. The masses are acting in the X- and Z-directions. The Y-direction is not relevant because there is no excitation load in this direction.
Now we calculate the first four eigenvalues of the model and verify that our model is stable. We obtain the lowest natural frequencies which we will then compare with the induced frequency to find out if a time history analysis is required.

<table>
<thead>
<tr>
<th>Mode No.</th>
<th>Eigenvalue $\lambda$ [1/s$^2$]</th>
<th>Angular Frequency $\omega$ [rad/s]</th>
<th>Natural frequency $f$ [Hz]</th>
<th>Natural period $T$ [s]</th>
<th>Induced frequency $f_i$ [Hz]</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1131,516</td>
<td>33,638</td>
<td>5,354</td>
<td>0,187</td>
<td>5,500</td>
</tr>
<tr>
<td>2</td>
<td>1216,551</td>
<td>34,879</td>
<td>5,551</td>
<td>0,180</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>1255,068</td>
<td>35,426</td>
<td>5,638</td>
<td>0,177</td>
<td></td>
</tr>
<tr>
<td>4</td>
<td>1576,375</td>
<td>39,703</td>
<td>6,319</td>
<td>0,158</td>
<td></td>
</tr>
</tbody>
</table>

The table above shows that the values of the natural frequencies are close to the induced frequency. Therefore, a time history analysis is needed for this model to exclude any resonance cases.

After the calculation of the eigenvalues, we open the RF-DYNAM Pro module again. To create the time diagram, we activate the Time History Analysis in the General tab of RF-DYNAM Pro (a Forced Vibrations license is needed to create a time history analysis). By selecting the Time diagram checkbox, we get the additional Time Diagram tab.

We will use the Periodic option to create a time diagram. For this type of time diagram, we use the period of the machine and a multiplier. The multiplier $k$ is set to 1 caused by the load in Load Cases 3 and 4 which is the maximum load.

$$\omega = 2 \cdot \pi \cdot f_i = 34.558 \cdot \frac{1}{\sigma}$$
We create two time diagrams, one for the vertical induced load of the engine and one for the horizontal induced load of the engine. In the diagram for the horizontally induced load, we take a phase shift of $\pi/2$. If the maximum vertical load is reached, the horizontal load at this time point $\pi/2$ is zero.
After defining the time diagram, we click the Dynamic Load Case tab to create a new Dynamic Load Case if one does not already exist. In the General sub-tab, we select the Direct Integration for the time history analysis type.

In the Time History Analysis sub-tab, we create two loading sets. In the first set, we select the time diagram TD1 in combination with Load Case 3. In the second set, we select the time diagram TD2 in combination with Load Case 4. These sets are important for the calculation because they are the link between the load cases and time diagrams.
Under the integration parameters, the time step is set manually to 0.001 seconds and the maximum time is set to 10 seconds. If the **Automatically according to time diagram** option is selected, the program will automatically define the time step and the maximum time for the calculation in accordance to the time diagram. In this example we will use 10 seconds as the maximum time to view the influence of the damping and resonance effects.

In the third sub-tab **Damping**, we define the structural damping $D$ of the steel frame model as 0.01 (Lehr’s damping).

After the input we calculate the model again by using the [OK & Calculate] button.
The first three eigenvalues are quite close to the excitation frequency. This means that the deformations and support reactions calculated by the time step analysis are very high.

For a better overview, we take a look at the support forces in the Z-direction by using the Time Course Monitor. This monitor gives an overview of the results on several points over the time, as well as a comparison between several points on the structure.

Support reaction of support node 10:
Deformation of node 49:

The frequency of the excitation case is close to the natural frequency of the structure. Due to this, the deformations and support forces are very high. This is not a true resonance which normally is caused by small differences in the excitation frequency and natural frequency but rather is caused by a numerical problem called “beating phenomenon.” This phenomenon occurs because of an inadequate numerical accuracy in the calculation (reference: J. W. Tedesco “Structural Dynamics” Section 5.2). This effect is typical with Finite Element calculations.

In the case that there is a resonance case, there are several options to prevent this from occurring. For our model, there are two simple options. We can either increase the stiffness or reduce the stiffness of the structure.
Reducing the Stiffness

To prevent the resonance case, we reduce the stiffness of the structure by modifying the cross-sections of the beams and columns.

Cross-sections after modifying the stiffness:

After changing the stiffness with the alternate cross-sections, we recalculate the model. The analysis with RF-DYNAM Pro shows that there is no resonance in the model. We see the natural frequency of the model is much lower than the induced frequency of the engines. The shape of the natural frequency has not changed, however.

<table>
<thead>
<tr>
<th>Mode No.</th>
<th>Eigenvalue $\lambda$ [1/s²]</th>
<th>Angular Frequency $\omega$ [rad/s]</th>
<th>Natural frequency $f$ [Hz]</th>
<th>Natural period $T$ [s]</th>
<th>Induced frequency $f_i$[Hz]</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>534,201</td>
<td>23,1128</td>
<td>3,679</td>
<td>0,272</td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>592,215</td>
<td>24,3355</td>
<td>3,873</td>
<td>0,258</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>1348,848</td>
<td>36,7267</td>
<td>5,845</td>
<td>0,171</td>
<td>5,500</td>
</tr>
<tr>
<td>4</td>
<td>1606,021</td>
<td>40,0752</td>
<td>6,378</td>
<td>0,157</td>
<td></td>
</tr>
</tbody>
</table>
When we take a look at the Time Course Monitor and compare the result before and after modifying the sections, we see that the deformation and the support forces are much smaller now.

Support reaction of support node 10 after modifying the stiffness:

Deformation of node 49 after modifying the stiffness:

For existing structures, there is the problem that a stiffness reduction is not possible. Changing cross-sections to a reduced size in a building is complicated. Therefore, the only options that exist would be to change the machine (alternate drive) or to increase the stiffnesses of the existing cross-sections.

The next steps are to export the results of the model to a result combination. Here the result combination can be combined with the self-weight. With this new result combination, it is possible to perform a stress or stability design of the structure. These topics are beyond the scope of this introductory example.