KISSsys

New features 2017

KISSsoft AG
Topics

1. Modelling
   - Gear stages library
   - Shaft split
   - Casing sizing

2. Templates
   - Efficiency
   - Load spectrum
   - Auxiliary results
   - Settings

3. Specific functions
   - System reliability
   - XML export
   - Reference point for thermal expansion
   - Shafts sizing

4. Handling
   - Renaming
   - 2D plot interface
   - Load factors
   - Power split
   - FEM carrier deformation
   - Forced response / Housing stiffness
Modelling

- Gear stages library
Modelling

- Shaft split

![Diagram of mechanical shaft and components with modeling software interface]

**Split operation**

Position on shaft \( [Y \text{ axis local}] \): 60 mm

[OK] [Cancel]
Modelling

- Casing sizing

![Diagram of casing sizing](image)

**Properties of Casing**

- **Type**: Cylinder
- **Length direction**: Y-axis
- **Position along X-axis**: 0.0000 mm
- **Position along Y-axis**: 0.0000 mm
- **Position along Z-axis**: 0.0000 mm
- **Diameter**: 3150.0000 mm
- **Length**: 2247.0000 mm
- **Width**: 3150.0000 mm
- **Height**: 3150.0000 mm
**Templates**

- **Efficiency**
  - Temperature drop over the cooler

![Image of the Set Power of the Oil Cooler dialog box with Rated cooling power of the cooler set to 200 W and Temperature drop over the cooler set to 3.]

- Journal bearing losses

![Image of a table showing various losses calculations in GearBox. The table includes columns for Losses, Calculated value [W], Correction factor, and Current value [W]. The values range from 0 to 4144.5 W.]

---

KISSSOFTWARE
Templates

- **Load spectrum**
  - KHb and Kgam inputs for each bin
  - Shaft temperature inputs for each bin

![Load spectrum template](image)

![Load distribution coefficient Kgam template](image)

![Temperature for load spectrum template](image)

![Define Load Spectrum template](image)

![User Interface table](image)

![Load Spectrum table](image)
Templates

- Load spectrum

  - Automatic consideration of additional columns
Templates

- **Load spectrum**
  - Modal analysis and contact analysis for each bin
  - Type of calculation update and possibility to export KISSsoft files for each selection
Templates

- **Load spectrum**
  - Automatic export of selected Load Spectrum values (DefineAdditionalData)
  - Possibility to assign the values to user defined Excel files
Templates

- **Auxiliary results**
  - New design for each element type
  - New price and torsion calculations

<table>
<thead>
<tr>
<th>Reference boundary</th>
<th>_O(Inout)</th>
<th>Calc Settings</th>
<th>Price Settings</th>
</tr>
</thead>
<tbody>
<tr>
<td>Calculate All</td>
<td>Gears</td>
<td>Bearings</td>
<td>Shafts</td>
</tr>
<tr>
<td></td>
<td>3.1388</td>
<td>0.952</td>
<td>2.783</td>
</tr>
<tr>
<td>Calculate Mass</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>[kg]</td>
<td>3.1388</td>
<td>0.952</td>
<td>2.783</td>
</tr>
<tr>
<td>Calculate Cost</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>[EUR]</td>
<td>62.7762</td>
<td>95.2</td>
<td>18.6335</td>
</tr>
<tr>
<td>Calculate Inertia</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>(kg·m²)</td>
<td>1077.4514</td>
<td>111.5567</td>
<td>332.1497</td>
</tr>
<tr>
<td>Calculate Kinetic Energy</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>[kJ]</td>
<td>0.2365</td>
<td>0.0245</td>
<td>0.0728</td>
</tr>
<tr>
<td>Calculate Backlash</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>[deg]</td>
<td>0.5399/0.2935</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>Calculate Torson</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>[deg]</td>
<td>0.0357</td>
<td>x</td>
<td>0.2161</td>
</tr>
</tbody>
</table>

- New settings to select calculations to consider
- New inputs for system required service life and reliability

<table>
<thead>
<tr>
<th>RATING</th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>System required service life [h]</td>
<td>20000</td>
<td>Disconnected</td>
<td>Connect</td>
</tr>
<tr>
<td>System required reliability [%]</td>
<td>99</td>
<td>Disconnected</td>
<td>Connect</td>
</tr>
<tr>
<td>Application factor KA</td>
<td>1.25</td>
<td>Disconnected</td>
<td>Connect</td>
</tr>
</tbody>
</table>

- Possibility to add or delete variables in an automated way

**Add new variable**

- Variable name: AmbTemp
- Table entry name: Ambient temperature

**Delete variable**

- Variable name: AmbTemp
Specific functions

- System reliability

![System reliability diagram and table](image)
Specific functions

- XML export

```
<!-- general KISSsys interface -->
  <root>
    <version>1.0</version>
    <units/>
    <shaft_calculation>
      <id>0</id>
      <shaft>
        <name>Shaft1</name>
        <global_origin/>
        <x-axis_vector/>
        <y-axis_vector/>
        <z-axis_vector/>
        <properties/>
        <bearing/>
        <bearing/>
        <gear/>
      </shaft>
    </shaft_calculation>
  </root>
```
Specific functions

- System reference point for thermal expansion

![Diagram showing specific settings and reference points for thermal expansion]
Specific functions

- Shafts sizing
Handling

- Inspect, Rename, Move objects in the tree
Handling

- 2D plot interface
Handling

▪ Mesh load factor
Handling

- Power split
Handling

- FEM carrier deformation

To calculate the bolt's misalignment, input details about the type of web in KISSsoft. In KISSsoft, first run a calculation (with the total sign) to ensure your data is consistent. Then enter your data about the web in the following screen:
- "Coefficients" tab -> in the "Face load factor" combination field: Axis alignment button
- "Axis alignment, proportional" tab -> in the "Planet bolt support" combination field, select "Perform FEM carrier calculation"
- "Details" button ->Conditions I / Conditions II

After you have filled the Conditions window with data, click on the "Calculate" button to start the FEM calculation.

The values for dr_p and dt_p are displayed as a result.

- Then the values for dr_p and dt_p are automatically recalculated on KISSsys side when the load is changing.
Handling

- **Forced response / Housing stiffness**

  - KISSsys forced response analysis includes now also TE and gear mesh stiffness variation as dynamic forces (on top of the unbalance forces of 2016 version).

  - OptiStruct FEM solver (from ALTAIR / HyperWorks) can now export reduced stiffness matrix in the KISSsys format. This reduced stiffness matrix can be used from housing stiffness calculation. Hence, the stiffness matrix formats supported are:

    i. KISSsys format: **ANSYS** (CADFEM commands), **OptiStruct** (directly).
    
    ii. Native: **ABAQUS, NASTRAN, Code_Aster**.
Thank you!
Any Questions?

KISSsoft AG
Reseller Meeting
April 2017