CONTRIBUTORS

Sandeep Vijayakar Ph.D., Hilliard OH
Karthikeyan Marambedu, Hilliard OH
Brett Baker, Hilliard OH
June 3, 2021
CONTENTS

List of Figures xi
List of Tables xxix
Preface xxxi

1 Introduction 1

2 A Coupled Spiral Bevel and Planetary Gear System 3
   2.1 The Input Rotor 3
      2.1.1 The Shaft 4
      2.1.2 The Spiral-Bevel Pinion Tooth 10
   2.2 The Intermediate Rotor 11
      2.2.1 The Shaft 11
      2.2.2 The Spiral-Bevel Gear Tooth 14
      2.2.3 The Spiral-Bevel Gear Rim 14
      2.2.4 The Sun Gear Tooth 16
   2.3 The Output Rotor 19
      2.3.1 The Carrier 19
      2.3.2 The Planetary Pinion 23
         2.3.2.1 The Planetary Pinion Teeth 23
         2.3.2.2 The Carrier Holes 23
         2.3.2.3 The Planetary Pinion Pin 26
         2.3.2.4 The Planetary Pinion Bearing 26
         2.3.2.5 The Planetary Pinion Shaft 26
         2.3.2.6 The Planetary Pinion Groups 26
      2.3.3 The Output Rotor Shaft 31
CONTENTS

2.4  The Reaction Rotor  
2.4.1  The Shaft  
2.4.2  The Ring Tooth and Rim  
2.4.3  The Ring Gear Spline Connector  

2.5  Pairs  

2.6  Setting up the analysis  

2.7  Analysis Results  
2.7.1  IGlass Postprocessing: Stress Contours, Deformation, Contact Grid, and Contact Pattern  
2.7.2  Contact Pressure  
2.7.3  Ring Gear Hoop Bending Stress  
2.7.4  Planetary Pinion Load Sharing  
2.7.5  Fatigue Analysis  

3  Automotive Rear Axle Assembly  

3.1  The Housing Model  

3.2  The Carrier Rotor  
3.2.1  The Carrier  
3.2.1.1  The Ring-Carrier Rotor Shaft  
3.2.1.2  The Straight Bevel Pinion  
3.2.1.3  The Hypoid Ring Gear  

3.3  The Propeller Shaft Rotor  
3.3.0.1  The Hypoid Pinion  
3.3.0.2  The Shaft  

3.4  The Half-Shafts  
3.4.0.1  The Bevel Gear  
3.4.0.2  The Shaft  

3.5  Connectors  
3.5.1  Journal Bearing  
3.5.2  Stiffness Bearing  
3.5.3  Roller Bearing  

3.6  Contact Surface Pairs  

3.7  The Analysis Setup  

3.8  Results  
3.8.1  Model Results with Stiffness Bearings  
3.8.2  Model Results with Roller Bearings  
3.8.3  EPGAlpha Results and Explanation  

4  Wind Turbine Gearbox  

4.1  The Housing Model  

4.2  The Ring Rotor  

4.3  The High Speed Pinion Rotor  

4.4  The Intermediate Rotor  

4.5  The Hollow Shaft Rotor  

4.6  The Planetary Sun Rotor  

4.7  The Planetary Carrier Rotor  
4.7.1  The Shaft
<table>
<thead>
<tr>
<th>Section</th>
<th>Title</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>4.7.2</td>
<td>The Carrier</td>
<td>139</td>
</tr>
<tr>
<td>4.7.3</td>
<td>The Pinions</td>
<td>144</td>
</tr>
<tr>
<td>4.8</td>
<td>The Bushing Rotors</td>
<td>149</td>
</tr>
<tr>
<td>4.9</td>
<td>Connectors</td>
<td>151</td>
</tr>
<tr>
<td>4.9.1</td>
<td>Roller Bearings</td>
<td>156</td>
</tr>
<tr>
<td>4.9.2</td>
<td>Stiffness Bearings</td>
<td>157</td>
</tr>
<tr>
<td>4.9.3</td>
<td>Spline Connector</td>
<td>158</td>
</tr>
<tr>
<td>4.10</td>
<td>Pairs</td>
<td>159</td>
</tr>
<tr>
<td>5</td>
<td>Simple Ball Bearing-Shaft Model</td>
<td>161</td>
</tr>
<tr>
<td>5.1</td>
<td>The Input Rotor</td>
<td>164</td>
</tr>
<tr>
<td>5.2</td>
<td>The Ball Bearing Connector</td>
<td>164</td>
</tr>
<tr>
<td>5.3</td>
<td>Analyzing the Ball Bearing-Shaft Model</td>
<td>167</td>
</tr>
<tr>
<td>6</td>
<td>A Double-Helical Planetary System</td>
<td>171</td>
</tr>
<tr>
<td>6.1</td>
<td>The Sun Rotor</td>
<td>171</td>
</tr>
<tr>
<td>6.2</td>
<td>The Ring Rotors</td>
<td>177</td>
</tr>
<tr>
<td>6.3</td>
<td>Ring Rotor Shafts</td>
<td>181</td>
</tr>
<tr>
<td>6.4</td>
<td>The Carrier Rotor</td>
<td>187</td>
</tr>
<tr>
<td>6.5</td>
<td>Connectors</td>
<td>193</td>
</tr>
<tr>
<td>6.6</td>
<td>Pairs</td>
<td>195</td>
</tr>
<tr>
<td>6.7</td>
<td>Analysis Setup</td>
<td>197</td>
</tr>
<tr>
<td>6.8</td>
<td>Results</td>
<td>199</td>
</tr>
<tr>
<td>7</td>
<td>Helical Reduction System with Housing</td>
<td>211</td>
</tr>
<tr>
<td>7.1</td>
<td>The Input Rotor</td>
<td>213</td>
</tr>
<tr>
<td>7.2</td>
<td>The Idler Rotor</td>
<td>220</td>
</tr>
<tr>
<td>7.3</td>
<td>The Output Rotor</td>
<td>225</td>
</tr>
</tbody>
</table>
11.1 Transmission3D Model Setup

11.1.1 The Pinion and Gear Rotors
- 11.1.1.1 The Pinion and Gear Rotor shafts
- 11.1.1.2 The Pinion and Gear Teeth

11.1.2 The Input Rotor

11.1.3 Condensed Housing

11.1.4 Connectors

11.1.5 Pairs

11.1.6 Setting Up and Running the Dynamic Analysis
- 11.1.6.1 Analysis Results Files
- 11.1.6.2 Choosing the Torsional Stiffness Value

11.2 Setting Up the Coustyx Model

12 Bolted Connection Example

12.1 Transmission3D Model Setup

12.1.1 The Carrier Rotor
- 12.1.1.1 The Carrier
- 12.1.1.2 The Carrier Rotor Shafts
- 12.1.1.3 The Carrier Sun Gear

12.1.2 The Input Rotor
- 12.1.2.1 Input Rotor Shaft
- 12.1.2.2 Input Rotor Sun Gear

12.1.3 Contact Pairs

13 5 Speed Manual Transmission Example

13.1 The Input Rotor
- 13.1.1 The Shaft
- 13.1.2 The Sun Gear

13.2 The Countershaft Rotor
- 13.2.1 The Countershaft Rotor Shafts
- 13.2.2 The Countershaft Rotor Sun Gears

13.3 The Driven Rotors
- 13.3.1 1st Speed Driven Rotor
- 13.3.2 2nd Speed Driven Rotor
- 13.3.3 3rd Speed Driven Rotor
- 13.3.4 4th Speed Driven Rotor
- 13.3.5 5th Speed Driven Rotor
- 13.3.6 The Reverse Rotors
  - 13.3.6.1 The Reverse Driven Rotor
  - 13.3.6.2 The Reverse Idler Rotor

13.4 The Output Shaft Rotor

13.5 The Reverse Idler Pin Rotor

13.6 Housing

13.7 Connectors

13.8 Pairs

13.9 Analysis Setup
14 Rear Axle Training

14.1 Step 1: Modeling the Hypoid Gear Pair
14.1.1 The Ring-Carrier Rotor
14.1.1.1 The Hypoid Ring Gear
14.1.1.2 The Hypoid Gear Rim
14.1.1.3 The Ring Carrier Rotor Shaft
14.1.2 The Propshaft Rotor
14.1.2.1 The Propshaft Rotor Shaft
14.1.2.2 The Hypoid Pinion and Rim
14.1.3 Pairs

14.2 Step 2: Importing the FE Carrier
14.2.1 The Carrier Menu
14.2.2 The Ring-Carrier Rotor Shaft

14.3 Step 3: Adding Roller Bearings to the Hypoid Pinion and Gear Rotors
14.3.1 The Ring Carrier Rotor Connector Races
14.3.2 The Propshaft Rotor Connector Races
14.3.3 The Connectors Inputs

14.4 Step 4: Modeling the Bevel Planetary Set w/Bearing, and Half Shaft Rotors w/Stiffness Support Bearings
14.4.1 The Ring-Carrier Rotor with Bevel Pinions
14.4.2 The Bevel PINION
14.4.2.1 The Bevel Pinion RACE
14.4.2.2 The Bevel Pinion WASHER
14.4.2.3 The Bevel Pinion TOOTH
14.4.2.4 The Bevel Pinion RIM
14.4.3 The Half-Shafts
14.4.3.1 The Shafts
14.4.3.2 The Bevel Gear
14.4.4 The Stiffness Connectors
14.4.5 The Bevel Pinion-Half Shaft Gear Pairs
14.4.6 Using Spherical Carrier Races and Shafts to Model Side Bevel-Washer Contact
14.4.6.1 The Carrier Rotor
14.4.6.2 The Bevel Rotors
14.4.6.3 Contact Pressure on Spherical Surface

14.5 Step 5: Importing the FE Housing Model
14.5.1 Updated Connector Menu Inputs

14.6 The Analysis Setup
14.7 Results
14.7.1 EPGAlpha Results and Explanation

15 Rear Axle Housing Acoustic Model

15.0.1 Transmission3D Dynamic Analysis Setup
15.0.1.1 Determination of DELTATIME and NTIMESTEPS Parameters
15.0.1.2 OP2 File Output Setup
15.0.1.3 Final Analysis Run and Results
15.0.2  *Coustyx* Acoustic Model Setup 477

16  Calculation of Lubricant Film Thickness & Flash Temperature 485
   16.1 Introduction 485
   16.2 Model 485
   16.3 Calculating Flash temperature and Film Thickness 485
   16.4 Validation of Results 485

17  Dynamic Condensation of a Thin Plate Using Craig-Bampton Method in *Hypermesh/Optistruct* and *Abaqus* 497
   17.1 Overview 497
      17.1.1 Dynamic Condensation of a Structure with *Hypermesh/Optistruct* 497
      17.1.1.1 Calculating Modal Frequencies of Condensed Structure from *Transmission3D* 501
      17.1.2 Model Setup and Substructuring Analysis with *Abaqus* 502
      17.1.2.1 Comparison with Theoretical Modal Frequencies 508

18  Plotting Dynamic Analysis Response 509
   18.1 Setting up the FEProbes 509
   18.2 Plotting FEProbe Time and Frequency Response 509

19  Bearing Defect Dynamics 513
   19.1 Model Setup 514
      19.1.1 Measured Signal 517
      19.1.2 Race Modeling 518
      19.1.3 Time Step 519
   19.2 Bearing State Tool 519
   19.3 Baseline model 520
   19.4 Bearing Defect Studies 521

20  Shaft Segment Point Load 529
   20.1 Defining a Point Load on a Shaft Segment 529
   20.2 Using a Point Load as a Sensor 529

21  Condensed Housing Master Node Load-Displacement 533
   21.1 Defining a Displacement Boundary Condition at a Master Node 533
   21.2 Defining a Load Boundary Condition at a Master Node 536

22  Hypoid Gear Pressfit Rainflow Fatigue Postprocessing 539
   22.1 Max Damage Fatigue Post-processing 539

References 543
LIST OF FIGURES

2.1 The coupled spiral bevel and planetary system. 4
2.2 The rotor menu for the input rotor. 5
2.3 Hypoid pinion shaft details. 6
2.4 The shaft segment menu for segment 1 of the input rotor. 7
2.5 The hypoid menu for the spiral-bevel pinion tooth mesh. 8
2.6 Hypoid pinion tooth details. 9
2.7 The rotor menu for the intermediate rotor. 11
2.8 Intermediate shaft details. 12
2.9 The shaft segment menu for segment 2 of the intermediate rotor. 13
2.10 The hypoid menu for the spiral-bevel gear tooth mesh. 14
2.11 Details of the spiral-bevel gear tooth on the intermediate rotor. 15
2.12 Details of the rim of the spiral-bevel gear on the intermediate rotor. 15
2.13 The rim menu for the spiral-bevel gear. 16
2.14 Details of the sun gear tooth on the intermediate rotor. 17
2.15 The tooth menu for the sun gear tooth on the intermediate rotor. 18
2.16 Details of the carrier attached to the output rotor. 19
2.17 The rotor menu for the output rotor. 20
2.18 The menu for the carrier on the output rotor. 21
2.19 The race menu for the FE carrier.
2.20 The pinion main menu for the output rotor.
2.21 Details of the pinion of the carrier on the output rotor.
2.22 The pinion tooth menu for the output rotor.
2.23 The carrier holes menu.
2.24 The output pinion pin shaft details.
2.25 The segment menu for segment 2 of the output pinion pin shaft.
2.26 The spherical double-row output pinion bearing.
2.27 The geometry menu for the spherical double-row output pinion bearing.
2.28 The output pinion shaft details.
2.29 The segment menu for segment 2 of the output pinion shaft.
2.30 The pinion groups menu.
2.31 Output Rotor Shaft Details.
2.32 Details of the ring tooth, ring rim, and shaft segment on the reaction rotor.
2.33 The rotor menu for the reaction rotor.
2.34 The ring tooth menu for the reaction rotor.
2.35 The ring gear spline connector details.
2.36 The ring gear spline connector menu.
2.37 The pairs menu.
2.38 The analysis setup menu.
2.39 The range menu.
2.40 Stress contours on the system.
2.41 The stress contours and deformation on the planetary system gears.
2.42 Contact grid on the spiral-bevel pinion.
2.43 Contact grid on the sun gear at the pinion 1 contact.
2.44 Contact grid on the ring gear at the pinion 1 contact.
2.45 The contact postprocessing menu.
2.46 Contact pressure on the ring gear teeth.
2.47 Contact pressure on the sun gear teeth.
2.48 The searchstress postprocessing menu.
2.49 Bending hoop stress in the fillet region of the ring gear teeth.
2.50 The bearing reaction postprocessing menu.
2.51 Fatigue postprocessing menu for pinion 1 side 1.
2.52 Critical point stress plot.
2.53 Modified Goodman diagram.
2.54 Fatigue damage contour plot.
2.55 Cumulative damage on pinion 1-side 1.
2.56 Cumulative damage on pinion 1-side 2.
2.57 Cumulative damage on pinion 2-side 1.
2.58 Cumulative damage on pinion 2-side 2.
2.59 Cumulative damage on pinion 3-side 1.
2.60 Cumulative damage on pinion 3-side 2.
2.61 Cumulative damage on pinion 4-side 1.
2.62 Cumulative damage on pinion 4-side 2.
2.63 Cumulative damage on the sun gear-side 2.
2.64 Cumulative damage on the sun gear-side 2.
2.65 Cumulative damage on the ring gear-side 1.
2.66 Cumulative damage on the ring gear-side 2.
3.1 A cut-away view of an automotive rear-axle assembly.
3.2 A schematic drawing of the automotive rear-axle assembly.
3.3 The housing finite element mesh.
3.4 The housing menu.
3.5 Housing race locations and dimensions.
3.6 Housing race menu for race 1.
3.7 Rotor 1, the carrier.
3.8 The rotor menu for the carrier rotor.
3.9 The carrier finite element mesh.
3.10 The carrier on rotor 1. \( z_e \) is the \( z \) coordinate measured with respect to the carrier origin, and \( z \) is the \( z \) coordinate measured.
3.11 The carrier menu.
3.12 The carrier race menu for the conical hypoid gear race.
3.13 The rotor 1 shaft SEGMENT menu.
3.14 The pinion menu.
3.15 The straight bevel differential pinion.
3.16 The pinion tooth menu.
3.17 The rim of the straight bevel differential pinion.
3.18 The bevel pinion washer.
3.19 The carrier holes menu. 85
3.20 The hypoid ring gear on rotor 1. 85
3.21 The hypoid menu for the carrier’s hypoid gear. 86
3.22 The rim of the hypoid ring gear on rotor 1. 86
3.23 Rotor 2, the prop-shaft. 87
3.24 The rotor menu for the prop-shaft rotor. 88
3.25 The hypoid pinion tooth. 89
3.26 The hypoid pinion menu. 89
3.27 The hypoid pinion rim. 90
3.28 Rotor 3, the left wheel half-shaft. 91
3.29 Rotor 4, the right wheel half-shaft. 92
3.30 The rotor menu for the left wheel rotor. 93
3.31 The rotor menu for the right wheel rotor. 94
3.32 The bevel gear of rotors 3 and 4. 95
3.33 The menu for a bevel gear of rotors 3 and 4. 96
3.34 The menu for a bevel gear of rotors 3 and 4. 96
3.35 The rim of the bevel gear of rotors 3 and 4. 97
3.36 Connector menu for journal bearing. 98
3.37 Stiffness Bearing HOUSING_WHEEL1 99
3.38 The menu for Stiffness Bearing 100
3.39 Cylindrical bearing on propshaft rotor 101
3.40 The GEOMETRY menu for Cylindrical Bearing 102
3.41 The GEOMETRY menu for PINIONHEADBRG Connector 102
3.42 Inputs of PINIONHEAD bearing 103
3.43 Hypoid pinion-gear pair menu. 103
3.44 The analysis setup menu 104
3.45 Stress contours on the hypoid pinion. 105
3.46 Contact Pattern on the hypoid pinion. 106
3.47 Stress contours on the hypoid gear. 107
3.48 Stress contours on one of the straight bevel pinions. 108
3.49 Stress contours on the right half shaft straight bevel gear. 109
3.50 Stress contours on the housing. 110
3.51 Stress contours on the pinion shaft with roller bearings. 111
3.52 Stress contours on the carrier with roller bearings.
3.53 Stress contours on the housing with roller bearings.
3.54 Stress contours on the housing with roller bearings.
3.55 The E,P,G,Alpha sign convention for a left-handed gear.
3.56 The E,P,G,Alpha sign convention for a left-handed gear.
3.57 The E,P,G,Alpha sign convention for a right-handed gear.
3.58 The E,P,G,Alpha sign convention for a right-handed gear.
4.1 The wind turbine gearbox assembly.
4.2 The wind turbine housing model.
4.3 Races on Housing - 1
4.4 Races on Housing - 2
4.5 The planetary ring rotor
4.6 The ring gear tooth menu.
4.7 The high speed pinion rotor.
4.8 The high speed pinion rotor shaft details.
4.9 The sun gear tooth menu.
4.10 The intermediate rotor.
4.11 The intermediate rotor shaft details.
4.12 The 1st sun gear tooth menu.
4.13 The 2nd sun gear tooth menu.
4.14 The hollow shaft rotor.
4.15 The hollow shaft rotor shaft details.
4.16 The sun gear tooth menu.
4.17 The planetary sun rotor.
4.18 The planetary sun rotor shaft details.
4.19 The sun gear tooth menu.
4.20 The planetary carrier rotor
4.21 The carrier rotor shaft details.
4.22 The carrier menu.
4.23 The carrier race definitions.
4.24 The carrier pinion assembly.
4.25 The carrier pinion menu.
4.26 The carrier pin.
4.27 The carrier pinion bearings. 146
4.28 The carrier pinion bearing geometry menu. 146
4.29 The carrier pinion shaft. 147
4.30 The carrier pinion tooth menu. 148
4.31 The housing bushing rotors. 149
4.32 The bushing shaft. 150
4.33 The roller bearing locations. 151
4.34 The stiffness bearing locations. 152
4.35 The stiffness bearing locations. 153
4.36 The stiffness bearing locations. 155
4.37 The CONNECTOR menu for the spline connector. 158
5.1 The Ball-Bearing Shaft FE Model. 162
5.2 The edit menu. 163
5.3 The rotor menu. 164
5.4 Input Rotor Shaft 165
5.5 The ball bearing geometry menu. 166
5.6 Geometry menu parameter details. 166
5.7 The rotor menu with applied force at rotor origin. 167
5.8 The analysis setup menu. 168
5.9 The range menu. 168
5.10 The connector contact grid menu. 169
5.11 The maximum principle normal stress on the system. 169
5.12 The contact grid on the ball bearing. 170
6.1 The double-helical planetary system. 172
6.2 The Sun rotor. 172
6.3 The sun rotor menu. 173
6.4 The sun rotor shaft 1 details. 174
6.5 The sun rotor shaft 2 details. 174
6.6 The segment menu for segment 1 of shaft 1. 175
6.7 The sun gear details for sun 1. 175
6.8 The tooth menu for sun gear 1. 176
6.9 The Ring1 rotor menu. 177
6.10 The Ring2 rotor menu. 178
6.11 The ring gear details.
6.12 The ring gear tooth menu.
6.13 The ring gear tooth menu.
6.14 The ring2 rotor rim shafts using rim FE mesh.
6.15 The ring2 rotor rim FE mesh model.
6.16 The shaft menu for shaft 2.
6.17 The segment menu for segment 1 of shaft 2.
6.18 The rotor 3 carrier menu using the FECARRIER_NASTRAN carrier option.
6.19 The shaft 1 segment menu.
6.20 The shaft 2 segment menu menu.
6.21 The Carrier rotor.
6.22 The Carrier rotor shaft details.
6.23 The Carrier rotor shaft segment menu.
6.24 The carrier pinion details.
6.25 The carrier pinion menu.
6.26 The carrier pinion pin menu.
6.27 The carrier pinion tooth menu.
6.28 The connector menu for connector 1.
6.29 The connector menu for the pinion journal bearing.
6.30 The pair menu for gear pair 1.
6.31 The pair menu for the shaft-gear contact.
6.32 The analysis setup menu.
6.33 The analysis range menu.
6.34 The contact pressure on the ring1 rotor rim.
6.35 The point displacement postprocessing menu.
6.36 Finding the element number and coordinates in IGlass.
6.37 The radial deformation on the ring1 rotor gear.
6.38 The tangential deformation on the ring 1 rotor gear.
6.39 The axial deformation on the ring1 rotor gear.
6.40 The point strain postprocessing menu.
6.41 Ring 1 rim sector 1.
6.42 The nodes at which the strain data is collected.
6.43 The element coordinate system.
<table>
<thead>
<tr>
<th>Figure</th>
<th>Description</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>6.44</td>
<td>The tangential strain on the ring1 rotor rim.</td>
<td>209</td>
</tr>
<tr>
<td>7.1</td>
<td>The Helical Reduction FE Model.</td>
<td>212</td>
</tr>
<tr>
<td>7.2</td>
<td>The Input Rotor.</td>
<td>213</td>
</tr>
<tr>
<td>7.3</td>
<td>The input rotor menu.</td>
<td>214</td>
</tr>
<tr>
<td>7.4</td>
<td>The input shaft details.</td>
<td>214</td>
</tr>
<tr>
<td>7.5</td>
<td>The input shaft menu.</td>
<td>215</td>
</tr>
<tr>
<td>7.6</td>
<td>The input shaft segment 1 menu.</td>
<td>216</td>
</tr>
<tr>
<td>7.7</td>
<td>The sun gear details.</td>
<td>217</td>
</tr>
<tr>
<td>7.8</td>
<td>The sun gear menu.</td>
<td>218</td>
</tr>
<tr>
<td>7.9</td>
<td>The sun gear tooth menu.</td>
<td>219</td>
</tr>
<tr>
<td>7.10</td>
<td>The Idler Rotor.</td>
<td>220</td>
</tr>
<tr>
<td>7.11</td>
<td>The idler shaft details.</td>
<td>221</td>
</tr>
<tr>
<td>7.12</td>
<td>The idler shaft segment 4 menu.</td>
<td>221</td>
</tr>
<tr>
<td>7.13</td>
<td>The idler sun gear 1 details.</td>
<td>222</td>
</tr>
<tr>
<td>7.14</td>
<td>The sun gear 1 tooth menu.</td>
<td>223</td>
</tr>
<tr>
<td>7.15</td>
<td>The idler sun gear 2 details.</td>
<td>223</td>
</tr>
<tr>
<td>7.16</td>
<td>The sun gear 2 tooth menu.</td>
<td>224</td>
</tr>
<tr>
<td>7.17</td>
<td>The Output Rotor.</td>
<td>225</td>
</tr>
<tr>
<td>7.18</td>
<td>The output shaft details.</td>
<td>226</td>
</tr>
<tr>
<td>7.19</td>
<td>The output shaft segment 8 menu.</td>
<td>226</td>
</tr>
<tr>
<td>7.20</td>
<td>The output sun gear details.</td>
<td>227</td>
</tr>
<tr>
<td>7.21</td>
<td>The sun gear tooth menu.</td>
<td>228</td>
</tr>
<tr>
<td>7.22</td>
<td>The FE Housing.</td>
<td>229</td>
</tr>
<tr>
<td>7.23</td>
<td>The FE housing details.</td>
<td>230</td>
</tr>
<tr>
<td>7.24</td>
<td>The FE housing menu.</td>
<td>230</td>
</tr>
<tr>
<td>7.25</td>
<td>The FE housing race menu.</td>
<td>231</td>
</tr>
<tr>
<td>7.26</td>
<td>The condensed housing details.</td>
<td>232</td>
</tr>
<tr>
<td>7.27</td>
<td>Importing the Nastran FE model and locating the origin.</td>
<td>233</td>
</tr>
<tr>
<td>7.28</td>
<td>Create interface nodes.</td>
<td>233</td>
</tr>
<tr>
<td>7.29</td>
<td>Creating RBE2 rigid body elements.</td>
<td>234</td>
</tr>
<tr>
<td>7.30</td>
<td>Creating and applying the ASET load collector.</td>
<td>235</td>
</tr>
<tr>
<td>7.31</td>
<td>Creating and defining the CMSMETH load collector card.</td>
<td>236</td>
</tr>
<tr>
<td>7.32</td>
<td>Defining the output type and condensing the model.</td>
<td>237</td>
</tr>
</tbody>
</table>
7.33 The housing menu for the condensed housing model.
7.34 Node menu for node 1.
7.35 Node menu for node 5.
7.36 Connector menu for stiffness connector 1.
7.37 Roller bearing connector 2 details.
7.38 Connector menu for roller bearing connector 2.
7.39 Connector geometry menu for roller bearing connector 2.
7.40 The pairs menu.
7.41 The analysis setup menu.
7.42 The analysis range menu.
7.43 The contact pressure on the input gear for the three misalignment cases.
8.1 The ring gear model.
8.2 The ring gear rotor menu.
8.3 The ring gear tooth menu.
8.4 The ring gear shaft segment menu.
8.5 The rotor menu for shaft support 1.
8.6 The shaft segment menu for shaft support 1.
8.7 The pairs menu for ring gear shaft-support shaft 1 pair.
8.8 The tooth side numbering convention used in Transmission3D.
9.1 The simple FE carrier planetary model.
9.2 The sun rotor menu.
9.3 The sun rotor shaft menu.
9.4 The sun rotor shaft details.
9.5 The sun rotor sun menu.
9.6 The sun rotor sun tooth menu.
9.7 The ring rotor menu.
9.8 The ring rotor shaft menu.
9.9 The ring rotor shaft details.
9.10 The ring rotor ring menu.
9.11 The ring rotor ring tooth menu.
9.12 The carrier rotor menu.
9.13 The carrier rotor shaft menu.
9.14 The carrier rotor shaft details.
9.15 The FE carrier menu. 268
9.16 The FE carrier details. 269
9.17 The carrier pinion menu. 269
9.18 The carrier PINHOLES menu. 270
9.19 The carrier pin shaft details. 270
9.20 The carrier pinion bearing menu. 271
9.21 The carrier pinion roller bearing geometry menu. 272
9.22 The carrier pinion shaft details. 272
9.23 The carrier pinion deck menu. 273
9.24 The carrier pinion tooth menu. 274
9.25 The sun bearing connector menu. 275
9.26 The sun roller bearing geometry menu. 275
9.27 The carrier bearing connector menu. 276
9.28 The carrier roller bearing geometry menu. 276
9.29 The pairs menu. 277

10.1 Helical Gear Pair Model. 280
10.2 Misalignment definition - 3D view. 282
10.3 Misalignment definition - 2D view. 283
10.4 Misalignment File(HELICALMISALIGNMENT.DAT). 283
10.5 Contact Pressure Distribution of Base Model. 284
10.6 Contact Pressure Distribution of Corrected Model. 286

11.1 The single helical gear pair model. 290
11.2 The rotor menu. 291
11.3 The shaft menu. 292
11.4 The gear shaft details. 293
11.5 The pinion rotor shaft details. 294
11.6 The pinion and gear tooth menu. 295
11.7 The input rotor menu. 296
11.8 The input rotor shaft details. 297
11.9 The condensed housing model. 298
11.10 The housing menu. 299
11.11 The connectors menu for the input rotor-pinion rotor torsional stiffness bearing. 300
11.12 The pairs menu. 302
11.13 Gear $\theta_z$ Response with Varying $k_{\theta z}$.

12.1 The bolted connection model. 304

12.2 The carrier rotor menu. 308

12.3 The carrier race menu. 309

12.4 The carrier race details. 310

12.5 The carrier contact surface details. 311

12.6 The carrier contact surface details. 312

12.7 The carrier shaft 1 details. 313

12.8 The carrier shaft 2 segment menu. 314

12.9 The bolts hole pattern on carrier shafts 2 and 3. 315

12.10 The carrier shaft 4 details. 316

12.11 The carrier shaft 5 details. 317

12.12 The carrier sun gear tooth menu. 318

12.13 The input rotor menu. 319

12.14 The input rotor shaft details. 320

12.15 The input sun gear tooth menu. 321

12.16 The contact pairs menu for shaft-shaft contact. 322

12.17 The contact pairs menu for gear tooth contact. 323

13.1 The 5 Speed Manual Transmission Model. 326

13.2 Cut-away view of the model components (reverse idler not pictured). 327

13.3 The EDIT menu. 328

13.4 The input rotor. 329

13.5 The ROTOR menu for the Input Rotor. 330

13.6 The input rotor shaft details. 331

13.7 The input rotor sun gear. 332

13.8 The input rotor sun gear TOOTH menu. 333

13.9 The countershaft rotor. 334

13.10 The countershaft ROTOR menu. 335

13.11 The countershaft rotor shaft details. 335

13.12 The countershaft rotor shaft 1 details. 336

13.13 The countershaft rotor shaft 2 through 5 details. 337

13.14 The countershaft rotor shaft 1 menu. 338

13.15 The countershaft sun gear 1. 339
<table>
<thead>
<tr>
<th>Figure</th>
<th>Description</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>13.16</td>
<td>The countershaft sun gear 1 TOOTH menu.</td>
<td>340</td>
</tr>
<tr>
<td>13.17</td>
<td>The countershaft sun gear 2.</td>
<td>341</td>
</tr>
<tr>
<td>13.18</td>
<td>The countershaft sun gear 2 TOOTH menu.</td>
<td>342</td>
</tr>
<tr>
<td>13.19</td>
<td>The countershaft sun gear 3.</td>
<td>343</td>
</tr>
<tr>
<td>13.20</td>
<td>The countershaft sun gear 3 TOOTH menu.</td>
<td>344</td>
</tr>
<tr>
<td>13.21</td>
<td>The countershaft sun gear 4.</td>
<td>345</td>
</tr>
<tr>
<td>13.22</td>
<td>The countershaft sun gear 4 TOOTH menu.</td>
<td>346</td>
</tr>
<tr>
<td>13.23</td>
<td>The countershaft sun gear 5.</td>
<td>347</td>
</tr>
<tr>
<td>13.24</td>
<td>The countershaft sun gear 5 TOOTH menu.</td>
<td>348</td>
</tr>
<tr>
<td>13.25</td>
<td>The countershaft sun gear 6.</td>
<td>349</td>
</tr>
<tr>
<td>13.26</td>
<td>The countershaft sun gear 6 TOOTH menu.</td>
<td>350</td>
</tr>
<tr>
<td>13.27</td>
<td>The 1st speed driven rotor.</td>
<td>352</td>
</tr>
<tr>
<td>13.28</td>
<td>The 1st speed driven ROTOR menu.</td>
<td>353</td>
</tr>
<tr>
<td>13.29</td>
<td>The 1st speed driven rotor shaft details.</td>
<td>354</td>
</tr>
<tr>
<td>13.30</td>
<td>The 1st speed driven sun gear 1.</td>
<td>355</td>
</tr>
<tr>
<td>13.31</td>
<td>The 1st speed driven sun gear 1 TOOTH menu.</td>
<td>356</td>
</tr>
<tr>
<td>13.32</td>
<td>The 2nd speed driven rotor.</td>
<td>357</td>
</tr>
<tr>
<td>13.33</td>
<td>The 2nd speed driven ROTOR menu.</td>
<td>358</td>
</tr>
<tr>
<td>13.34</td>
<td>The 2nd speed driven rotor shaft details.</td>
<td>359</td>
</tr>
<tr>
<td>13.35</td>
<td>The 2nd speed driven sun gear 1.</td>
<td>360</td>
</tr>
<tr>
<td>13.36</td>
<td>The 2nd speed driven sun gear 1 TOOTH menu.</td>
<td>361</td>
</tr>
<tr>
<td>13.37</td>
<td>The 3rd speed driven rotor.</td>
<td>362</td>
</tr>
<tr>
<td>13.38</td>
<td>The 3rd speed driven ROTOR menu.</td>
<td>363</td>
</tr>
<tr>
<td>13.39</td>
<td>The 3rd speed driven rotor shaft details.</td>
<td>364</td>
</tr>
<tr>
<td>13.40</td>
<td>The 3rd speed driven sun gear 1.</td>
<td>365</td>
</tr>
<tr>
<td>13.41</td>
<td>The 3rd speed driven sun gear 1 TOOTH menu.</td>
<td>366</td>
</tr>
<tr>
<td>13.42</td>
<td>The 4th speed driven rotor.</td>
<td>367</td>
</tr>
<tr>
<td>13.43</td>
<td>The 4th speed driven ROTOR menu.</td>
<td>368</td>
</tr>
<tr>
<td>13.44</td>
<td>The 4th speed driven rotor shaft details.</td>
<td>369</td>
</tr>
<tr>
<td>13.45</td>
<td>The 4th speed driven sun gear 1.</td>
<td>370</td>
</tr>
<tr>
<td>13.46</td>
<td>The 4th speed driven sun gear 1 TOOTH menu.</td>
<td>371</td>
</tr>
<tr>
<td>13.47</td>
<td>The 5th speed driven rotor.</td>
<td>372</td>
</tr>
<tr>
<td>13.48</td>
<td>The 5th speed driven ROTOR menu.</td>
<td></td>
</tr>
</tbody>
</table>
13.49 The 5th speed driven rotor shaft details. 373
13.50 The 5th speed driven sun gear 1. 374
13.51 The 5th speed driven sun gear 1 TOOTH menu. 375
13.52 The reverse gear configuration. 376
13.53 The reverse driven ROTOR menu. 377
13.54 The reverse driven rotor shaft details. 378
13.55 The reverse driven sun gear 1. 379
13.56 The reverse driven sun gear 1 TOOTH menu. 380
13.57 The reverse idler ROTOR menu. 381
13.58 The reverse idler rotor shaft details. 382
13.59 The reverse idler sun gear 1. 382
13.60 The reverse idler sun gear 1 TOOTH menu. 383
13.61 The output shaft rotor. 384
13.62 The output shaft rotor menu. 385
13.63 The output shaft details. 386
13.64 The idler pin rotor. 387
13.65 The idler pin rotor menu. 388
13.66 The idler pin shaft details. 389
13.67 The finite element housing model. 390
13.68 The nodal housing constraints. 391
13.69 The housing menu. 391
13.70 The finite housing race locations. 392
13.71 The housing race menu. 393
13.72 The pairs menu. 397
14.1 A cut-away view of an automotive rear-axle assembly. 402
14.2 A schematic drawing of the automotive rear-axle assembly. 403
14.3 The hypoid gear pair model. 404
14.4 The ring carrier ROTOR menu. 405
14.5 The ring carrier rotor hypoid COMMON menu. 406
14.6 The ring carrier rotor hypoid gear details. 407
14.7 The ring carrier rotor hypoid gear rim details. 408
14.8 The ring carrier rotor hypoid gear rim menu. 409
14.9 The hypoid rim segment position definitions. 410
<table>
<thead>
<tr>
<th>Figure No.</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>14.10</td>
<td>The ring carrier rotor SHAFT menu.</td>
</tr>
<tr>
<td>14.11</td>
<td>The ring carrier rotor SHAFT details.</td>
</tr>
<tr>
<td>14.12</td>
<td>The propshaft ROTOR menu.</td>
</tr>
<tr>
<td>14.13</td>
<td>The propshaft rotor SHAFT menu.</td>
</tr>
<tr>
<td>14.14</td>
<td>The propshaft rotor shaft details.</td>
</tr>
<tr>
<td>14.15</td>
<td>The hypoid pinion COMMON menu.</td>
</tr>
<tr>
<td>14.16</td>
<td>The hypoid pinion details.</td>
</tr>
<tr>
<td>14.17</td>
<td>The hypoid pinion rim details.</td>
</tr>
<tr>
<td>14.18</td>
<td>The pairs menu for step 1 of the model.</td>
</tr>
<tr>
<td>14.19</td>
<td>The ROTOR menu for rotor 1 of step 2.</td>
</tr>
<tr>
<td>14.20</td>
<td>The CARRIER menu.</td>
</tr>
<tr>
<td>14.21</td>
<td>The carrier RACE menu details for Step 2.</td>
</tr>
<tr>
<td>14.22</td>
<td>The ring-carrier rotor shaft SEGMENT details for step 2.</td>
</tr>
<tr>
<td>14.23</td>
<td>The hypoid ring carrier race locations, step 3.</td>
</tr>
<tr>
<td>14.24</td>
<td>Rotor 2 shaft details w/bearing race locations.</td>
</tr>
<tr>
<td>14.25</td>
<td>The GEOMETRY menu details for connector 3.</td>
</tr>
<tr>
<td>14.26</td>
<td>The GEOMETRY menu for the TAPERED roller bearing type.</td>
</tr>
<tr>
<td>14.27</td>
<td>The rear axle model with step 4 additions.</td>
</tr>
<tr>
<td>14.28</td>
<td>The CARRIER menu.</td>
</tr>
<tr>
<td>14.29</td>
<td>The carrier PINION menu.</td>
</tr>
<tr>
<td>14.30</td>
<td>The PINRACE menu.</td>
</tr>
<tr>
<td>14.31</td>
<td>The contact WASHER menu.</td>
</tr>
<tr>
<td>14.32</td>
<td>The bevel pinion tooth details.</td>
</tr>
<tr>
<td>14.33</td>
<td>The bevel pinion TOOTH menu.</td>
</tr>
<tr>
<td>14.34</td>
<td>The bevel pinion rim.</td>
</tr>
<tr>
<td>14.35</td>
<td>The rotor menu for the left wheel rotor.</td>
</tr>
<tr>
<td>14.36</td>
<td>The rotor menu for the right wheel rotor.</td>
</tr>
<tr>
<td>14.37</td>
<td>Rotor 3 shaft details.</td>
</tr>
<tr>
<td>14.38</td>
<td>Rotor 4 shaft details.</td>
</tr>
<tr>
<td>14.39</td>
<td>Shafts 2 and 3 of the half shafts.</td>
</tr>
<tr>
<td>14.40</td>
<td>The bevel gear of rotors 3 and 4.</td>
</tr>
<tr>
<td>14.41</td>
<td>The menu for a bevel gear of rotors 3 and 4.</td>
</tr>
<tr>
<td>14.42</td>
<td>The menu for a bevel gear of rotors 3 and 4.</td>
</tr>
</tbody>
</table>
14.43 The CONNECTORS menu for the bevel pinion bearing. 446
14.44 The PAIRS menu. 447
14.45 The spherical race model assembly schematic. 448
14.46 The carrier rotor menu. 449
14.47 The carrier RACE menu. 449
14.48 The carrier races. 450
14.49 The spherical SHAFT menu. 451
14.50 The spherical shaft SEGMENT menu. 451
14.51 The spherical shaft details. 452
14.52 The bevel rotor details. 453
14.53 The bevel gear TOOTH geometry menu. 454
14.54 The contact pressure distribution at the spherical shaft-bevel shaft contact surface. 455
14.55 The housing finite element mesh. 456
14.56 The housing menu. 457
14.57 Housing race locations and dimensions. 458
14.58 Housing race menu for race 1. 459
14.59 The analysis setup menu 462
14.60 Stress contours on the pinion shaft with roller bearings. 463
14.61 Stress contours on the carrier with roller bearings. 464
14.62 Stress contours on the housing with roller bearings. 465
14.63 Stress contours on the housing with roller bearings. 466
14.64 The E,P,G,Alpha sign convention for a left-handed gear. 468
14.65 The E,P,G,Alpha sign convention for a left-handed gear. 468
14.66 The E,P,G,Alpha sign convention for a right-handed gear. 469
14.67 The E,P,G,Alpha sign convention for a right-handed gear. 469
15.1 The hypoid pinion dynamic time response $\theta_z$ component. 472
15.2 The hypoid gear dynamic time response $\theta_z$ component. 473
15.3 The steady state RANGE menu input parameters with OP2 file generation. 474
15.4 OP2 file setup from the OP2FILES menu. 475
15.5 Importing the structure into Coustyx from the OP2 results file. 477
15.6 Importing the external shrink wrap mesh in Coustyx. 478
15.7 Creating the skin and BE mesh. 478
15.8 Creating the structure velocity boundary condition. 479
<table>
<thead>
<tr>
<th>Figure</th>
<th>Description</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>15.9</td>
<td>Setting the structure velocity boundary condition.</td>
<td>480</td>
</tr>
<tr>
<td>15.10</td>
<td>Setting up the analysis sequence solver controls.</td>
<td>480</td>
</tr>
<tr>
<td>15.11</td>
<td>Loading the analysis sequence frequencies from the structure response.</td>
<td>481</td>
</tr>
<tr>
<td>15.12</td>
<td>The analysis sequence frequency table.</td>
<td>481</td>
</tr>
<tr>
<td>15.13</td>
<td>Setting up just in time loading of housing frequency response.</td>
<td>482</td>
</tr>
<tr>
<td>15.14</td>
<td>The 3D sound pressure contour iGlass results.</td>
<td>483</td>
</tr>
<tr>
<td>16.1</td>
<td>Gear Pair Model.</td>
<td>486</td>
</tr>
<tr>
<td>16.2</td>
<td>PATTERN menu</td>
<td>487</td>
</tr>
<tr>
<td>16.3</td>
<td>Flash Temperature Distribution w/Friction Coefficient = 0.048</td>
<td>488</td>
</tr>
<tr>
<td>16.4</td>
<td>Flash Temperature Distribution w/Autocomputed Friction Coefficient</td>
<td>489</td>
</tr>
<tr>
<td>16.5</td>
<td>Film Thickness Distribution w/Friction Coefficient = 0.048</td>
<td>490</td>
</tr>
<tr>
<td>16.6</td>
<td>Film Thickness Distribution w/Autocomputed Friction Coefficient</td>
<td>491</td>
</tr>
<tr>
<td>16.7</td>
<td>Comparison of Results.</td>
<td>493</td>
</tr>
<tr>
<td>16.8</td>
<td>Comparison of Results.</td>
<td>494</td>
</tr>
<tr>
<td>16.9</td>
<td>Comparison of Results.</td>
<td>495</td>
</tr>
<tr>
<td>16.10</td>
<td>Comparison of Results.</td>
<td>496</td>
</tr>
<tr>
<td>17.1</td>
<td>The <em>Hypermesh</em> plate dynamic condensation model setup.</td>
<td>498</td>
</tr>
<tr>
<td>17.2</td>
<td>Application of the ASET nodal constraints.</td>
<td>499</td>
</tr>
<tr>
<td>17.3</td>
<td>Application of the SPC nodal constraints.</td>
<td>500</td>
</tr>
<tr>
<td>17.4</td>
<td>Optistruct analysis panel menu.</td>
<td>500</td>
</tr>
<tr>
<td>17.5</td>
<td>The HOUSING menu inputs for the condensed housing type.</td>
<td>501</td>
</tr>
<tr>
<td>17.6</td>
<td>The initial load step boundary conditions.</td>
<td>503</td>
</tr>
<tr>
<td>17.7</td>
<td>Creating the frequency analysis step.</td>
<td>503</td>
</tr>
<tr>
<td>17.8</td>
<td>The frequency analysis step boundary condition.</td>
<td>504</td>
</tr>
<tr>
<td>17.9</td>
<td>Creating the substructure generation step.</td>
<td>504</td>
</tr>
<tr>
<td>17.10</td>
<td>The substructure generation step boundary condition.</td>
<td>505</td>
</tr>
<tr>
<td>17.11</td>
<td>Creating the job for the analysis.</td>
<td>506</td>
</tr>
<tr>
<td>17.12</td>
<td>Editing the keywords to create condensed massstiffness and recovery matrix output files.</td>
<td>507</td>
</tr>
<tr>
<td>18.1</td>
<td>The FEPROBES menu.</td>
<td>510</td>
</tr>
<tr>
<td>18.2</td>
<td>Time and frequency response of output probe data.</td>
<td>511</td>
</tr>
<tr>
<td>19.1</td>
<td>Sequence of steps in enveloping technique.</td>
<td>514</td>
</tr>
<tr>
<td>19.2</td>
<td>Two methods evaluated for envelope detection.</td>
<td>515</td>
</tr>
<tr>
<td>19.3</td>
<td>Simple system with one roller bearing.</td>
<td>516</td>
</tr>
</tbody>
</table>
19.4 Roller bearing load distribution from static analysis
19.5 Probes on the roller bearing
19.6 Effect of Circular Division on bearing outer race
19.7 Screenshot of bearing state tool.
19.8 Dynamic response of no defect model, before and after the modeling improvements.
19.9 Dynamic response of no defect model.
19.10 Dynamic response from the models with bearing pits
19.11 Envelope extraction from a signal with outer race defect
19.12 Frequency Spectrum of outer race defect signal without data processing
19.13 Envelope extraction from a signal with inner race defect
19.14 Frequency Spectrum of inner race defect signal without data processing
19.15 Location of inner race pits in the load zone contacting rollers
19.16 Envelope extraction from a signal with roller defect
19.17 Frequency Spectrum of roller defect signal without data processing
20.1 The shaft segment menu with point load enabled.
20.2 The shaft segment point load menu.
20.3 The local stress concentration at the point load location.
20.4 The contact pattern before applying the point load.
20.5 The contact pattern after applying the point load.
20.6 The CONCLOADS.DAT output file with displacement values at the point load location.
21.1 The condensed housing menu with retained node displacement enabled.
21.2 The condensed housing node menu with displacement.
21.3 Undeformed housing model (left) vs nodal displacement model (right) results.
21.4 The condensed housing menu with retained node load enabled.
21.5 The condensed housing node menu with force.
21.6 Unloaded housing model (left) vs nodal load model (right) results.
22.1 The modified rear axle model hypoid gear rim with press-fit at carrier.
22.2 The FATIGUE post-processing menu with MAX_DAMAGE criterion selected.
22.3 Critical point stress plot without press-fit.
22.4 Critical point stress plot with press-fit.
LIST OF TABLES

2.1 Planetary Pinion Angular Position 31
2.2 Pinion Load Sharing 51
2.3 Fatigue Cycle Data 56
3.1 FE Carrier Internal Race Menu Input Data 78
3.2 FE Carrier Cylindrical Bearing Race Input Data 79
3.3 Connector Locations and Orientations 98
3.4 Connector Stiffnesses 99
3.5 Tapered Roller Bearing 100
3.6 Contact Grid Parameters 101
3.7 E, P, G, α Deflection Values 115
4.1 The GROUP menu inputs 140
4.2 Connector Race Locations 154
4.3 Cylindrical Roller Bearing Geometry Menu Inputs 156
4.4 Tapered Roller Bearing Geometry Menu Inputs 156
4.5 Stiffness Bearing Stiffness Inputs 157
4.6 Pair Menu Inputs 159
6.1 FE Rim RACE Menu Input Data 181
6.2 The group menu inputs. 189
<table>
<thead>
<tr>
<th>Table</th>
<th>Description</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>6.3</td>
<td>The stiffness bearing connector race positions.</td>
<td>194</td>
</tr>
<tr>
<td>7.1</td>
<td>The connector origin and race positions.</td>
<td>240</td>
</tr>
<tr>
<td>7.2</td>
<td>The connector roller geometry parameters.</td>
<td>242</td>
</tr>
<tr>
<td>7.3</td>
<td>The roller bearing runout error.</td>
<td>243</td>
</tr>
<tr>
<td>8.1</td>
<td>Start and end angles for partial support shafts.</td>
<td>254</td>
</tr>
<tr>
<td>10.1</td>
<td>Gear Data</td>
<td>281</td>
</tr>
<tr>
<td>10.2</td>
<td>Contact Pattern on Gear 1</td>
<td>287</td>
</tr>
<tr>
<td>11.1</td>
<td>Condensed housing NODE menu inputs.</td>
<td>299</td>
</tr>
<tr>
<td>11.2</td>
<td>The connector origin and race positions.</td>
<td>301</td>
</tr>
<tr>
<td>11.3</td>
<td>Connector Stiffnesses</td>
<td>301</td>
</tr>
<tr>
<td>11.4</td>
<td>Connector Damping</td>
<td>301</td>
</tr>
<tr>
<td>11.5</td>
<td>Range menu parameters.</td>
<td>303</td>
</tr>
<tr>
<td>12.1</td>
<td>The FILE menu input parameters.</td>
<td>309</td>
</tr>
<tr>
<td>13.1</td>
<td>Countershaft sun gear axial positions.</td>
<td>336</td>
</tr>
<tr>
<td>13.2</td>
<td>Connector Race Locations</td>
<td>395</td>
</tr>
<tr>
<td>13.3</td>
<td>Connector Stiffnesses</td>
<td>396</td>
</tr>
<tr>
<td>13.4</td>
<td>Tapered Roller Bearing</td>
<td>396</td>
</tr>
<tr>
<td>14.1</td>
<td>FE Carrier Internal Race Menu Input Data</td>
<td>422</td>
</tr>
<tr>
<td>14.2</td>
<td>Connector Locations and Orientations</td>
<td>427</td>
</tr>
<tr>
<td>14.3</td>
<td>Tapered Roller Bearing</td>
<td>428</td>
</tr>
<tr>
<td>14.4</td>
<td>The half shaft connector origin and race positions.</td>
<td>444</td>
</tr>
<tr>
<td>14.5</td>
<td>Half Shaft Bearing Stiffness Inputs</td>
<td>445</td>
</tr>
<tr>
<td>14.6</td>
<td>The half shaft connector origin and race positions.</td>
<td>460</td>
</tr>
<tr>
<td>14.7</td>
<td>E, P, G, α Deflection Values</td>
<td>467</td>
</tr>
<tr>
<td>16.1</td>
<td>Gear Data</td>
<td>487</td>
</tr>
<tr>
<td>16.2</td>
<td>Pattern Menu Material/Fluid Properties</td>
<td>492</td>
</tr>
<tr>
<td>17.1</td>
<td>T3D NODE menu inputs.</td>
<td>501</td>
</tr>
<tr>
<td>17.2</td>
<td>Transmission3D vs Theoretical Natural Frequencies - Optistruct.</td>
<td>501</td>
</tr>
<tr>
<td>17.3</td>
<td>T3D NODE menu inputs.</td>
<td>508</td>
</tr>
<tr>
<td>17.4</td>
<td>Transmission3D vs Theoretical Natural Frequencies - Abaqus.</td>
<td>508</td>
</tr>
<tr>
<td>19.1</td>
<td>Simple bearing model - System information.</td>
<td>517</td>
</tr>
<tr>
<td>19.2</td>
<td>Bearing pit conditions</td>
<td>522</td>
</tr>
</tbody>
</table>
PREFACE

The Transmission3D computer program is the culmination of many years of work. During this time, we have received active support and encouragement from many people. We would especially like to thank Timothy Krantz of the Army Research Laboratory at the NASA Glenn Research Center for his support and encouragement. We are grateful for the support of the U.S. Army’s SBIR program, without which the development of this program would not have been possible.
CHAPTER 1

INTRODUCTION

The Transmission3D software package is capable of running all the three-dimensional parallel axis models described in the Planetary3D Validation Manual [2]. In addition, Transmission3D can also be used to model systems with non-parallel axes, with hypoid and bevel gears. This manual is intended to walk you through the details of setting a system with non-parallel axis.
CHAPTER 2

A COUPLED SPIRAL BEVEL AND PLANETARY GEAR SYSTEM

Helicopters usually have both spiral bevel and planetary gears in close proximity to each other, as shown in Figure 2.1. To the extent that light and flexible structures cause interactions between the spiral bevel set and the planetary set, it might become necessary to model the entire system.

The example described in this chapter shows you how to set up such a transmission. The example illustrates the use of a flexible carrier. Transmission3D is capable of modeling rigid carriers and housings as well, but these are often unrealistic assumptions. The inputs for this example can be loaded from the session file called CoupledSPBevelPlanetary.ses in the subdirectory SAMPLES/CoupledSPBevelPlanetary under the default working directory.

We use inches as the unit of length, and lbf as the unit of force in this example.

The model consists of four 'Rotors'. The input shaft with an attached spiral bevel pinion forms the first rotor. We shall refer to it as the 'Input Rotor'. The spiral bevel gear is connected to the sun gear of the planetary gear set. The two together form the second rotor: the 'Intermediate Rotor'. The planetary carrier forms the third rotor. It delivers the output power, so we call it the 'Output Rotor'. The fourth rotor is formed by the ring gear. We call it the 'Reaction Rotor'. The pinions are considered part of the carrier for the purpose of organizing the input data.

In Transmission3D, a rotor is considered to be of TYPE=INPUT if we specify its RPM. In this example, we need to specify the RPM of two members; the input rotor and the reaction rotor. So we have two rotors for which TYPE=INPUT. We choose to specify the load torque at the output rotor. Thus the TYPE is OUTPUT for this rotor. The intermediate rotor has a zero external torque, so it is of TYPE=IDLER. We could just as well have chosen to make this rotor to be of TYPE=OUTPUT, and set the external torque to be zero. It should be emphasized that whether TYPE=INPUT or OUTPUT has nothing to do with where the power flows into the system and out of it.

We have decided to locate the origin of the transmission at the intersection of the spiral bevel gear axes. We have also chosen to make this the origin of each of the four rotors.

2.1 The Input Rotor

The main rotor menu for the input rotor is shown in Figure 2.2. It is accessed by setting NROTORS=4 and ROTOR=1. Since the origin of the rotor coincides with the system origin, XPOSN, YPOSN and ZPOSN for the input rotor are all
zero. We have chosen to make the axis of this rotor to point along the Y axis of the system. Hence \( AX=0, AY=1 \) and \( AZ=0 \). Since we would like to specify the RPM of this rotor, its TYPE is INPUT. We specify \( RPM=-4000 \). A negative value for RPM implies that we are following the left hand rule about the rotor axis. (A positive value would mean that we use the right hand rule.)

The rotor consists of one shaft with a spiral-bevel pinion resting on two stiffness type connectors, as shown in Figures 2.1 and 2.3. So we have checked ENABLESHAFTS and ENABLEHYPOIDS, and set \( NSHAFTS=1 \) and \( NHYPOIDS=1 \).

Because the rotor is connected to ground through the stiffness type connectors, there is no need to directly constrain the rotor’s reference frame. Thus \( UXCONSTRAINT, UYCONSTRAINT, UZCONSTRAINT, THETAXCONSTRAINT \) and \( \Theta YCONSTRAINT \) are all left unchecked. Since these rotor degrees of freedom are unconstrained, we have the opportunity to apply external loads on these degrees of freedom. We choose not to do so, hence \( FX, FY, FZ, MX \) and \( MY \) are all zero.

2.1.1 The Shaft

Figure 2.3 shows the shaft geometry. The shaft is positioned at the correct distance from the rotor origin along the rotor axis by specifying \( AXIALPOSNSHAFT=2.6039800460 \).

The shaft has six segments. Segment 1 connects to the base surface of the spiral-bevel pinion so we make the outer diameter of this segment a ‘race’ surface by setting \( ODRACE=TRUE \). Figure 2.4 displays the inputs of the segment menu for shaft segment 1. Since the spiral-bevel pinion is generated by the \textit{HypoidFaceMilled} software package, the parameters that define the position of shaft segment 1 must be entered to an accuracy level at least as precise as that of the \textit{HypoidFaceMilled} output parameters \( RA, ZA, RB, \) and \( ZB \). The Fourier order of the race \( CIRCORDERROUTER \) is set to 4 and the polynomial order in the axial direction \( AXIALORDERROUTER \) is set to 2. These match the orders of interpolation used at the spiral-bevel base surface.

The outer diameter surfaces of segments 3 and 5 interface with stiffness type connectors. We set \( ODRACE=TRUE \) for these segments too, with \( CIRCORDERROUTER=2 \) and \( AXIALORDERROUTER=2 \) chosen for both segments. The shaft needs to be attached to the rotor’s reference frame, so that the shaft’s stiffness matrix is non singular. This is accomplished by constraining the outer diameter of segment 2. The external torque acting on the rotor flows through this constraint.
Figure 2.2: The rotor menu for the input rotor.
A COUPLED SPIRAL BEVEL AND PLANETARY GEAR SYSTEM

Figure 2.3: Hypoid pinion shaft details.
Figure 2.4: The shaft segment menu for segment 1 of the input rotor.
Figure 2.5: The hypoid menu for the spiral-bevel pinion tooth mesh.
Figure 2.6: Hypoid pinion tooth details.
2.1.2 The Spiral-Bevel Pinion Tooth

The Hypoid feature allows us to mount a gear tooth with arbitrary geometry on a rotor. We use a slightly altered version of the spiral-bevel gear pair that is the standard example in the HypoidFaceMilled software package [3, 4]. The alteration occurs on the base surface of the pinion gear. In this example, the base surface geometry is conical, whereas the standard example uses a cylindrical base surface. The pinioncalyx.msh and gearcalyx.msh HypoidFaceMilled output files can be found in the subdirectory SAMPLES/CoupledSPBevelPlanetary under the default working directory. This gear pair has a 12 tooth pinion mating with a 36 tooth gear. After generating the spiral-bevel gear set using the HypoidFaceMilled software, the name of the file containing the finite element mesh is entered into the Hypoid menu (Figure 2.5). The same base surface geometry (Figure 2.6) used in HypoidFaceMilled is entered into the RABASE, ZABASE, RBBASE and ZBBASE fields of this menu. These values are generated by HypoidFaceMilled and define the position of the pinion. It is recommended that these values be copied directly from the information window after generating the model in HypoidFaceMilled in order to ensure the same precision level. The CIRCORDER=4 and AXIALORDER=2 fields match the circular order and axial order of shaft segment 1 with which the pinion mates.

The orientation of the pinion with respect to the rotor axis orientation is controlled by the AXISDIRECTION field. We have selected SAME because the positive rotor axis points from the toe to the heel of the pinion.
2.2 The Intermediate Rotor

The intermediate rotor carries the spiral-bevel gear and the planetary sun gear, connected by a shaft, as shown in Figures 2.1 and 2.8. The main rotor menu for the intermediate rotor is shown in Figure 2.7. It is accessed by setting ROTOR=2 in the rotor menu.

The origin of this rotor also coincides with the system origin, so XPOSN, YPOSN and ZPOSN are again zero. We have chosen to make the axis of this rotor to point along the Z axis of the system. Hence AX=0, AY=0 and AZ=1. Since we would like to have no external torque applied to it, we set TYPE=IDLER.

The rotor consists of one shaft with a spiral-bevel gear and a helical sun gear, as shown in Figures 2.1 and 2.8. So we have checked ENABLESHAFTS and ENABLESUNS, and set NSHAFTS=1 and NSUNS=1.

We have decided not to use stiffness type connectors to constrain this rotor. Instead, we have directly applied constraints to the rotor’s reference frame. Thus UXCONSTRAINT, UYCONSTRAINT, UZCONSTRAINT, THETAXCONSTRAINT and THETAYCONSTRAINT are all checked. Since these rotor degrees of freedom are constrained, we have the opportunity to specify the displacements associated with these degrees of freedom. We choose not to do so, hence UX, UY, UZ, THETAX and THETAY are all zero.

2.2.1 The Shaft

The shaft of the intermediate rotor has three segments (Figure 2.8). The outer diameters of segments 1 and 3 have ODRACE checked in order to interface with the spiral-bevel gear and the sun gear, respectively. In order to keep the rotor stiffness matrix non-singular, we need to constrain the shaft somewhere. We decided to constrain the inner diameter of segment 2 by checking IDCONSTRAINED. We would like to apply the constraints only to the rigid body type of modes, and allow the deformation to be unconstrained. So we set IDTYPE to FLEXIBLE (Figure 2.9).
Section 2.8: Intermediate shaft details.
**Fig. 2.9:** The shaft segment menu for segment 2 of the intermediate rotor.
2.2.2 The Spiral-Bevel Gear Tooth

The Hypoid feature is used to import the spiral-bevel gear generated by the HypoidFaceMilled software package [3, 4]. The name of the finite element mesh file is entered into the Hypoid menu (Figure 2.10). The base surface geometry (Figure 2.11) is created with the RABASE, ZABASE, RBBASE and ZBBASE parameters. The number of teeth on the gear, NTEETH is 36. The CIRCORDER=4 and AXIALORDER=2 fields match the circular and axial orders of shaft segment 1 with which the gear mates.

The orientation of the gear with respect to the rotor axis orientation is controlled by the AXISDIRECTION field. We select SAME because the rotor axis is positive from the toe of the gear to its heel.

2.2.3 The Spiral-Bevel Gear Rim

The rim model provides the additional material between the base surface of the spiral-bevel gear and the shaft segment 1 as shown in Figure 2.12. The rim is made up of 4 segments. The R and Z coordinates for the points A and B are entered for each of the four segmentsminto the rim menu. Figure 2.13 shows the rim menu for segment 1.
Figure 2.11: Details of the spiral-bevel gear tooth on the intermediate rotor.

Figure 2.12: Details of the rim of the spiral-bevel gear on the intermediate rotor.
2.2.4 The Sun Gear Tooth

Figure 2.14 shows the geometry of the 27 tooth sun gear on the intermediate rotor. We have decided not to have a rim model for this sun gear. Instead, the inner diameter (INNERDIA=2.25) directly mates with the outer diameter of segment 3 of the rotor shaft. Figure 2.15 shows the sun gear tooth menu.
Figure 2.14: Details of the sun gear tooth on the intermediate rotor.
Figure 2.15: The tooth menu for the sun gear tooth on the intermediate rotor.
2.3 The Output Rotor

The output rotor consists of the flexible carrier and a small one-segment shaft as shown in Figures 2.1 and 2.16. The main rotor menu for the output rotor is shown in Figure 2.17. It is accessed by setting ROTOR=3 in the rotor menu.

The origin of this rotor coincides with the system origin, so XPOSN, YPOSN and ZPOSN are again zero. The axis of this rotor points along the Z axis of the system. Hence AX=0, AY=0 and AZ=1. Since we would like to apply an external torque to it, we set TYPE=OUTPUT. We set TORQUE=-40000. The sign of this torque is negative, so the sense of the torque follows the left hand rule about the rotor axis.

The rotor consists of one shaft with a flexible carrier. So we have checked ENABLESHAFTS and ENABLECARRIERS, and set NSHAFTS=1 and NCARRIERS=1.

We have directly applied constraints to the rotor’s reference frame, so UXCONSTRAINT, UYCONSTRAINT, UZCONSTRAINT, THETAXCONSTRAINT and THETAYCONSTRAINT are all checked. We set UX, UY, UZ, THETAX and THETAY to zero.

2.3.1 The Carrier

Figure 2.18 shows the carrier menu for the output rotor. The carrier in this example is set to the FECARRIER_NASTRAN carrier TYPE. The FILE and RACE sub-menus appear when this carrier type is selected. NFECARRIERFILES sets the number of FE mesh files to be imported. The FEFILENAME, SUBTREENAME, and PREFERREDCUTTINGDIR are entered within the FILE menu.

NODETOLERANCE sets the tolerance used to search for nodes of race and contact surfaces. In this example, we set NODETOLERANCE=0.0005. The REMOVEOPTIONALNODES flag provides us with the option to ignore the mid-nodes of quadratic elements. In this case, we set it to FALSE. MAXJOINTANGLE is set to to automatically...
Figure 2.17: The rotor menu for the output rotor.
**Figure 2.18:** The menu for the carrier on the output rotor.

**Figure 2.19:** The race menu for the FE carrier.
smooth any joining edges forming an angle less than 15 degrees. We also set AXIALSHIFT=0 to align the carrier reference frame with the reference frame of the output rotor.

The carrier contains pinions, so the ENABLEPINIONS box is checked. The carrier contains four identical copies of one pinion, so we set NPINIONS=1 and NGROUPS=4. We choose not to use cyclic symmetry in this example, so USESECTORALSYMMETRY is left unchecked.

The number of connecting surfaces in this example is set within the NRACES field to 1. The RACE menu contains the inputs necessary to define the connecting surfaces. A minimum one race is required when using the FECARRIER_NASTRAN carrier type since the carrier must be connected to a constrained shaft. The race menu inputs for the race in this example are shown in Figure 2.19.
2.3.2 The Planetary Pinion

The planetary pinions data is grouped with the carrier data for convenience. The pinion submenu (Figure 2.20) of the carrier menu provides access to this data. The carrier has four identical copies of one pinion. So we have NPINIONS=1. NPINIONS can be greater than 1, for instance, in a compound planetary system. The axis of the planetary pinion shown in Figure 2.21 is at a distance of 3.50 from the carrier axis. So RADPOSN is set to 3.50. The nominal angular position of the first pinion is THETAPOSN=0.0 Degrees. The output pinion pin and shaft are set to COMPOUND. This allows us to model the pin and pinion shaft with multiple segments. The output pinion has just one deck, so NDECKS is 1.

2.3.2.1 The Planetary Pinion Teeth

The tooth submenu of the pinion shown in Figure 2.22 is accessed from the pinion menu. The midface section of the pinion tooth is at a distance of +5.00 from the rotor origin (as shown in Figure 2.21), so AXIALPOSN is 5.00. The PHASEANGLE is irrelevant because we only have one deck. NTEETH is set to 35.

2.3.2.2 The Carrier Holes

The CARRIERHOLES menu, shown in Figure 2.23, defines the race surfaces at the carrier hole(s) where the pinion pin connects. The pin in this example is supported on both ends so we leave ISCANTILEVEREDPINION=FALSE. The back and front hole radii are entered into the BACKRADIUS and FRONTRADIUS...
Figure 2.21: Details of the pinion of the carrier on the output rotor.
Figure 2.22: The pinion tooth menu for the output rotor.
The rotor axis is positive from the back to front side of the carrier. \( Z_1, Z_2, Z_3, \) and \( Z_4 \) are the axial distances from the carrier origin, entered in ascending order.

### 2.3.2.3 The Planetary Pinion Pin

The output pinion pin (Figure 2.24) connects to the flexible carrier and the inner race of the pinion bearing. The pin shaft is modeled with \( \text{NSEGMENTS}=3 \) and the first segment begins at \( \text{AXIALPOSNSHAFT}=4.231 \). Segments 1 and 3 require \( \text{ODRACE} \) to be checked in order to connect the pin to the carrier. \( \text{ODRACE} \) is checked on segment 2 to attach the pin to the inner race of the pinion bearing. Figure 2.25 shows the segment menu for the second segment of the pin.

### 2.3.2.4 The Planetary Pinion Bearing

The pinion bearing (Figure 2.26) interfaces with the pinion pin and pinion shaft. The pinion bearing is a double-row spherical roller bearing, which allows us to model the detailed contact at the rolling element surfaces. The bearing is modeled in the PINION menu under the bearing sub-menu. The \( \text{AXIALPOS}=5.00 \) places the bearing origin at the mid-face of the output pinion. Choosing \( \text{TYPE}=\text{ROLLERS} \) activates the sub-menus for a roller type connector. The geometry menu, shown in Figure 2.27 is used to enter the geometric parameters of the bearing. Within the cage menu, \( \text{AUTOCOMPUTE}=\text{TRUE} \) is selected to automatically model the cage based on the design parameters entered into the geometry menu.

### 2.3.2.5 The Planetary Pinion Shaft

The output pinion shaft (Figure 2.3) connects to the outer race of the pinion bearing and the base surface of the output pinion gear. The shaft is modeled in the pinionshaft sub-menu within the pinion menu. The \( \text{AXIALPSONSHAFT}=4.275 \) and \( \text{NSEGMENTS}=2 \). Segment one is a small shaft segment, modeled for the purpose of applying a flexible shaft constraint. This constraint fixes the outside diameter of the output pinions to their reference frame. The segment menu for the second shaft segment is given in Figure 2.29. Note, the order of interpolation on the inner and outer races of this segment must match the order of the interfacing race on the pinion bearing and output pinion base surface, respectively.

### 2.3.2.6 The Planetary Pinion Groups

The pinion group submenu (Figure 2.30) of the carrier menu controls the number and locations of the copies of the planetary pinion. We have 4 identical copies, so \( \text{NGROUPS}=4 \). The angular
Figure 2.24: The output pinion pin shaft details.

Figure 2.25: The segment menu for segment 2 of the output pinion pin shaft.
A COUPLED SPIRAL BEVEL AND PLANETARY GEAR SYSTEM

Inside diameter race interfaces with output pinion pin
AXIALORDEROUTER=2
CIRCORDEROUTER=4

Outside diameter race interfaces with output pinion shaft
AXIALORDEROUTER=2
CIRCORDEROUTER=4

Figure 2.26: The spherical double-row output pinion bearing.

Figure 2.27: The geometry menu for the spherical double-row output pinion bearing.
Figure 2.28: The output pinion shaft details.
Figure 2.29: The segment menu for segment 2 of the output pinion shaft.

Figure 2.30: The pinion groups menu.
Table 2.1: Planetary Pinion Angular Position

<table>
<thead>
<tr>
<th>Pinion ID</th>
<th>Angular Position (deg)</th>
<th>m</th>
</tr>
</thead>
<tbody>
<tr>
<td>Pinion 1</td>
<td>0.0000</td>
<td>0</td>
</tr>
<tr>
<td>Pinion 2</td>
<td>91.4286</td>
<td>32</td>
</tr>
<tr>
<td>Pinion 3</td>
<td>180.0000</td>
<td>63</td>
</tr>
<tr>
<td>Pinion 4</td>
<td>271.4286</td>
<td>95</td>
</tr>
</tbody>
</table>

positions of the planetary pinions are determined by the number of teeth of the sun and ring gears by the following equation:

$$\Theta = \frac{m}{N_s + N_r} \times 360$$

(2.1)

where: $m$ = an integer

$N_s$ = Number of Teeth on Sun = 27

$N_r$ = Number of Teeth on Ring = 99

$\Theta$ = Angular Position of Pinion (deg)

Table 2.1 provides the parameters used in Equation 2.1 and the corresponding angular pinion positions used as inputs of the GROUP menu.

2.3.3 The Output Rotor Shaft

Figure 2.31 provides the details for the output rotor shaft. The shaft is required in order for the rotor to contain a shaft constraint. In order to do so we model the shaft with one segment, which connects to the front of the carrier, and we constrain the outer diameter of the shaft to the rotor reference frame by checking ODRACE. The interface between the front of the carrier and the back of the shaft is created by checking ENABLEBACKINTERFACE in the shaft menu. We also define a race on the front of the carrier in the CVTBDF conversion of the carrier.
Figure 2.31: Output Rotor Shaft Details.
2.4 The Reaction Rotor

The reaction rotor consists of the planetary’s internal ring gear and a small two-segment shaft as shown in Figures 2.1 and 2.32.

The main rotor menu for this rotor is shown in Figure 2.33. It is accessed by setting ROTOR=4 in the rotor menu. The origin of this rotor also coincides with the system origin, so XPOSN, YPOSN and ZPOSN are again all zero. The rotor axis points along the Z axis of the system. Hence AX=0, AY=0 and AZ=1. Since we would like to keep this rotor stationary, we make TYPE=INPUT and set RPM=0.

The rotor consists of one shaft and one ring gear. So we have checked ENABLESHAFTS and ENABLERINGS, and set NSHAFTS=1 and NRINGS=1.

We have directly applied constraints to the rotor’s reference frame, so UXCONSTRAINT, UYCONSTRAINT, UZCONSTRAINT, THETAXCONSTRAINT and THETAYCONSTRAINT are all checked. We set UX, UY, UZ, THETAX and THETAY to zero.
Figure 2.33: The rotor menu for the reaction rotor.
2.4.1 The Shaft

The shaft model on the reaction rotor is used to tie the ring gear teeth to the spline connector and to apply the nodal constraint of the reaction rotor. Accordingly, the shaft has two segments, one which connects the ring gear base surface to the spline, and another small segment on which the nodal constraint is applied. The shaft has an axial position of AXIALPOSNSHAFT=4.8 with respect to the rotor’s origin. The inside diameter of segment 1 is constrained and the reaction torque flows in from this diameter.

2.4.2 The Ring Tooth and Rim

The ring gear tooth data is entered into the tooth submenu shown in Figure 2.34. The ring gear has NTEETH=99 teeth. Figure 2.34 gives the INNERDIA, ROOTDIA, and OUTERDIA for the ring gear. All race surfaces on the reaction rotor use an AXIALORDER=2 and CIRCORDER=64.

2.4.3 The Ring Gear Spline Connector

The ring gear spline connector (Figure 2.35) is modeled in the connectors sub-menu within the edit menu. The spline connector origin is coincident with the rotor origin (XPOS=0, YPOS=0, ZPOS=0) and we set the rotational axis in the same direction as the reaction rotor (AX=0, AY=0, AZ=1). The spline interfaces with segment two of the reaction rotor shaft, so we set AXIALPOSNS1RACE1 = AXIALPOSITION1RACE2 = 4.9 and AXIALPOSNS2RACE1 = AXIALPOSNS2RACE2 = 5.1. The spline TYPE = EXTERNALSPLINED and NSPLINES=24. The order of interpolation of the spline surface at its interfaces are set automatically. Figure 2.36 shows the connector menu for the spline with the remaining inputs.
Figure 2.35: The ring gear spline connector details.

Figure 2.36: The ring gear spline connector menu.
2.5 Pairs

The pairs menu is used to define the contact pairs in the model. Pairs can consist of gear tooth contact pairs or pairs between surfaces chosen to be modeled using a contact constraint. In this example we use only the prior. There are three gear pairs in the model so we set NPAIRS=3. Since we have only one pinion group, the sun-pinion and ring-pinion contacts are each modeled using one pair. The spiral-bevel gear pair is the third pair. Figure 2.37 shows the pairs menu and its inputs for the ring-pinion gear pair, with TYPE=RINGPINION. SEPTOL is the maximum separation of the mating surfaces to be used to search for contacting nodes. Any nodes at a distance greater than this value are not used for the contact pair. Here we set SEPTOL=0.003037 inches. NPROFDIVS and NFACEDIVS are the number of divisions on each side of the mid-face point of the gear in the profile and face directions, respectively. We set both to 3 in this case. ADAPTIVEGRID=ON automatically sets the spacing of these divisions in the profile direction.

2.6 Setting up the analysis

The analysis setup inputs are shown in Figures 2.38 and 2.39. The analysis is written to the file postproc.dat by setting POSTPROCWRI=TRUE. NTIMESTEPS is set to 11 in the range menu with DELTATIME=0.000212. DELTATIME is calculated such that a single tooth on one of the planetary gear components passes through a full mesh.
Figure 2.38: The analysis setup menu.
cycle in 11 time steps. The following equations are used to calculate the time for one tooth mesh cycle ($\Delta T_{cycle}$) and DELTATIME between each time step ($\Delta t$).

$$\Delta T_{cycle} = \frac{2\pi}{(\omega_{sun} - \omega_{carrier}) \times N_{sun}}$$

$$\Delta t = \frac{T_{cycle}}{NTIMESTEPS - 1}$$

Setting SPLITPOSTPROCFILE=TRUE stores the data for each time step to a separate data file. We also set INITIALTIME=0.00.

2.7 Analysis Results

2.7.1 IGlass Postprocessing: Stress Contours, Deformation, Contact Grid, and Contact Pattern

Figure 2.40 displays the maximum principle normal stresses on the system. The carrier and intermediate shaft components are partially hidden for clarity purposes. Figure 2.41 shows the deformation of the planetary gears.

Figures 2.42 through 2.44 show the contact grids for the hypoid pinion, sun gear and ring gear, respectively. The sun and ring gear contact grids are given for the pinion 1-sun and pinion 1-ring gear contact pairs.

The results presented in this section are obtained by creating a postprocessing IGlass file. For more information on IGlass files, refer to [5].
Figure 2.40: Stress contours on the system.
Figure 2.41: The stress contours and deformation on the planetary system gears.
Figure 2.42: Contact grid on the spiral-bevel pinion.
Figure 2.43: Contact grid on the sun gear at the pinion 1 contact.
Figure 2.44: Contact grid on the ring gear at the pinion 1 contact.
2.7.2 Contact Pressure

The contact postprocessing menu is displayed in Figure 2.45. The inputs given are for the sun-pinion contact pair. This contact occurs on tooth numbers 19 through 22 of the sun gear over the course of one mesh cycle. We search the entire tooth surface for the contact pressure, so we set SPROFBEGIN, SPROFEND, TFACEBEGIN, and TFACEEND accordingly. The contact pressure data is output to the file ContactPressure_SunGear.dat. This file is located in the subdirectory SAMPLES/CoupledSPBevelPlanetary under the default working directory (a similar file for the ring gear is available as well). Also within this subdirectory are the contact postprocessing script files, which can be used to generate the .dat files for the sun and ring gear contact pressures at the four planetary pinions. The script files are programmed to change the contact postprocessing menu inputs for each contact point on the sun and ring gears, then store the pressure data to the output data file of the corresponding gear. The MATLAB program PlotContactPressure.m is also located here and can be executed to generate the contact pressure plots for each gear. This program retrieves the data from each of the output files and plots the contact pressure at each pinion contact over the course of one mesh cycle. The resulting plots are given in Figures 2.46 and 2.47.
Figure 2.46: Contact pressure on the ring gear teeth.
Contact Pressure−SunGear

Figure 2.47: Contact pressure on the sun gear teeth.
2.7.3 Ring Gear Hoop Bending Stress

The search stress menu (Figure 2.48) is used to generate the bending stress data for the gear teeth. We demonstrate here how to use this menu to plot the hoop bending stress in the fillet region of the ring gear teeth. In order to do so, we would like to generate the maximum and minimum bending stresses for both sides of each ring gear tooth. Figure 2.48 provides the inputs required for the maximum stress on side 1 of the ring gear. We are interested in the maximum principle normal stress component, so we set COMPONENT = MAXPPLSTRESS. Choosing XAXIS = TIME generates stress data for each time instant of the selected gear teeth. In order to generate the data in the most time effective manner, we run the searchstress menu for one timestep and one tooth at a time. So, we have BEGINSTEP = ENDSTEP = TOOTHBEG = TOOTHEND = 1. Choosing SURFACE = FILL_SURFRING_1_1 selects the fillet region of tooth side 1 as the search region. SPROFBEGIN, SPROFEND, TFACEBEGIN, and TFACEND are the tooth surface coordinates in the profile and facewidth directions which define the search area. These inputs are left to their default values for this example, which are set automatically when the tooth SURFACE is selected. NUMSPROF and NUMTFACE set the number of sample points within the defined grid area in the profile and facewidth direction, respectively. NUMDEPTH is the minimum distance from a contact point at which a sample point will be used. This is used to filter out fictitious stress values associated with edge contact and is nominally set to a value of one-quarter of the tooth height.

The script files SEARCHSTRESS_MAXPPLSTRESS_SCRIPT-RingGearSide*.dat and SEARCHSTRESS_MINPPLSTRESS_SCRIPT-RingGearSide*.dat are located in the subdirectory SAMPLES/CoupledSBevelPlanetary under the default working directory. The SEARCHSTRESS_MAXPPLSTRESS_SCRIPT- RingGearSide1.dat script file executes the searchstress menu, one time step at a time, for the maximum principal normal bending stress on side 1 of each ring gear tooth. SEARCHSTRESS_MAXPPLSTRESS_SCRIPT- RingGearSide2.dat does the same for side 2. SEARCHSTRESS_MINPPLSTRESS_SCRIPT-RingGearSide1.dat and SEARCHSTRESS_MINPPLSTRESS_SCRIPT-RingGearSide2.dat set the menu inputs and generate the results for the minimum principle normal stresses.

The four search stress scripts generate two data files (RingGearSearchStressSide1.dat and RingGearSearchStressSide2.dat) one for each tooth side. The output files are ordered with the maximum principle normal stress scripts executed first, followed by the minimum principle normal scripts. These output files are also available in the SAMPLES/CoupledSBevelPlanetary subdirectory for the user’s reference. The MATLAB program PlotRingGearHoopStress.m is also located in this folder and can be used to generate the plot of ring gear hoop stress shown in Figure 2.49. The program is written to read the output of the two data files, compare the two tooth sides for maximum and minimum stress values, and plot stress vs. tooth number.
Figure 2.48: The searchstress postprocessing menu.
Figure 2.49: Bending hoop stress in the fillet region of the ring gear teeth.
Figure 2.50: The bearing reaction postprocessing menu.

Table 2.2: Pinion Load Sharing

<table>
<thead>
<tr>
<th>Gear ID</th>
<th>Force (lbs)</th>
<th>Load Sharing</th>
</tr>
</thead>
<tbody>
<tr>
<td>Pinion 1</td>
<td>-2814.98</td>
<td>0.2463</td>
</tr>
<tr>
<td>Pinion 2</td>
<td>-2870.44</td>
<td>0.2512</td>
</tr>
<tr>
<td>Pinion 3</td>
<td>-2862.30</td>
<td>0.2504</td>
</tr>
<tr>
<td>Ring</td>
<td>-2881.36</td>
<td>0.2521</td>
</tr>
<tr>
<td>Total</td>
<td>-11429.08</td>
<td>1.0000</td>
</tr>
</tbody>
</table>

2.7.4 Planetary Pinion Load Sharing

The bearing reaction postprocessing menu (Figure 2.50) is used to obtain the reaction forces at the planetary pinion bearings. The planetary pinion load sharing can be determined by finding the bearing reactions in the tangential direction and calculating the percentage of the total load carried by each individual pinion. The bearing reaction menu shown is that of pinion 1, so BEARING = PINIONBRG_3.1.1_1.1. The tangential component of the reaction force is obtained by selecting COMPONENT = FY. Since the load sharing is constant for each time step we select BEGINSTEP = ENDSTEP = 1. Obtaining the reaction force for each of the remaining pinion bearings requires changing only the BEARING input to the desired pinion number. Table 2.2 displays the results for the planetary pinions in this example.

2.7.5 Fatigue Analysis

Transmission3D’s fatigue postprocessing menu (Figure 2.51) is used to generate the fatigue data for the Coupled Spiral-Bevel Planetary model. The fatigue menu output consists of three sets of data: critical point data, fatigue data, and fatigue life data. This data is automatically displayed in the figure window within Transmission3D in the form of a critical point stress diagram (Figure 2.52), Modified Goodman Diagram (Figure 2.53), and a fatigue damage plot.
Figure 2.51: Fatigue postprocessing menu for pinion 1 side 1.
Figure 2.52: Critical point stress plot.
Figure 2.53: Modified Goodman diagram.
Figure 2.54: Fatigue damage contour plot.
Table 2.3: Fatigue Cycle Data

<table>
<thead>
<tr>
<th>Gear ID</th>
<th>NCycles (30000 in-lbs)</th>
<th>NCycles (40000 in-lbs)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Pinion</td>
<td>100000</td>
<td>7500</td>
</tr>
<tr>
<td>Sun</td>
<td>1000000</td>
<td>75000</td>
</tr>
<tr>
<td>Ring</td>
<td>45000</td>
<td>3500</td>
</tr>
</tbody>
</table>

(Figure 2.54). The critical point stress diagram shows the maximum and minimum principle normal stresses over one full gear rotation. The critical point is the location which experiences the maximum equivalent pure alternating stress over the course of the cycle. The Modified Goodman Diagram plots the alternating stress vs. mean stress of the same critical point location. The red line represents the maximum equivalent stress. Any critical point data falling above the blue endurance limit line represents a point where fatigue damage occurs, while those points below this line will not experience damage. If a point were located above the green yield limit line, yield failure would be expected to occur. The fatigue damage plot shows the damage distribution on the selected tooth surface for a selected number of cycles. The damage is minimal in the provided plot since only one cycle is analyzed.

The user also has the option to save the generated data in the form of a data file to suit their specific postprocessing needs. Here we will demonstrate how to use the fatigue life data to calculate cumulative damage effects and plot the damage contours on the tooth surfaces for each of the planetary gears in the model. Figure 2.51 shows the inputs required to generate the data file for the drive side of planetary pinion group 1. We use the default fatigue properties suggested for steel and set CRITERION=EQUIV PURE ALTERN. Choosing BODY=OUTPUT CARRIER PINION1 GROUP1 selects pinion group 1, and we choose SURFACE=FILL SURF PIN 1 1 to return the data for the fillet region of the drive side. SPROFBEGIN, SPROFEND, TFACEBEGIN, and TFACEEND are left to their default values, which are set automatically when the fillet surface region is selected. NUMSPROF and NUMTFACE sets the number of sample points used to collect data in the profile and face directions, respectively. We are interested in the results on the tooth surface only, so DEPTHBEGIN=DEPTHEND=0 and NUMDEPTH=1. NTOOTHCYCLES=1 since we are only interested in the fatigue life data, which is independent of the number of cycles chosen.

Two postprocessing script files are included in the subdirectory SAMPLES/CoupledSPBevel Planetary under the default working directory. These scripts run the fatigue postprocessing menu for both sides of all of the gears in the planetary system by entering the inputs of 2.51 and then changing the BODY and SURFACE inputs for each gear and side. The two script files correspond to the two torque levels of interest (30,000 in-lbs and 40,000 in-lbs) and access separate session and postprocessing files for each torque value. Note, in each of these files, the results are saved to a separate data file for each gear and side selected.

A MATLAB program PlotCumulativeDamage.m is also included in the SAMPLES/CoupledSP BevelPlanetary subdirectory. This program retrieves the data from the results files output by the fatigue postprocessing menu, arranges the three data sets into separate matrices, and plots the cumulative damage. Comments are included within the m-file to guide the user through the program. Figures 2.55 through 2.66 show the resulting cumulative damage contours for the planetary gears at the the two torque levels. Table 2.3 provides the cycle data used to generate the plots.
Cumulative Damage Plot – Pinion1–Side1

Figure 2.55: Cumulative damage on pinion 1-side 1.
Figure 2.56: Cumulative damage on pinion 1-side 2.
Figure 2.57: Cumulative damage on pinion 2-side 1.
Figure 2.58: Cumulative damage on pinion 2-side.
Figure 2.59: Cumulative damage on pinion 3-side 1.
Figure 2.60: Cumulative damage on pinion 3-side 2.
Figure 2.61: Cumulative damage on pinion 4-side 1.
Figure 2.62: Cumulative damage on pinion 4-side 2.
Figure 2.63: Cumulative damage on the sun gear-side 2.
Figure 2.64: Cumulative damage on the sun gear-side 2.
Figure 2.65: Cumulative damage on the ring gear-side 1.
Figure 2.66: Cumulative damage on the ring gear-side 2.
An automotive rear axle is a complex gear system, consisting of a hypoid gear set and a straight bevel differential set (Figure 14.1). We are going to show how to model such a system. We will also show how to include detailed models of a flexible housing and carrier in the analysis. The inputs for this example can be loaded from the session file called RearAxle.ses in the subdirectory SAMPLES/RearAxle under the default working directory. The housing and carrier finite element mesh are also in this directory.

For this example, we have selected the Newton as the unit of force and millimeter as the unit of length.

A schematic drawing of a rear axle system is shown in Figure 14.2. The power comes into the system through the propeller shaft. A hypoid pinion mounted on this shaft mates with a hypoid ring gear mounted on the carrier. The carrier has four straight bevel pinions that mate with the straight bevel gears on two half shafts. One of these half shafts drives the left wheel and the other drives the right wheel.

This system has two degrees of freedom, i.e. two angular velocities have to be specified to completely determine the kinematics of the system. We would like to specify the RPM on the input propshaft and left half shaft. Transmission3D will compute the angular velocities for the remaining members. The externally applied torque can be specified for these members whose angular velocity is not specified. We apply non-zero torque to the right half shaft, and a zero external torque on the carrier.

3.1 The Housing Model

Following industrial practice, A CAD model of the housing was first created. We used Pro-E for this step, but any commercial CAD package could have been used. Most companies already go through this step for their drawing and manufacturing needs.

The CAD model was then converted into a Nastran finite element mesh using the commercially available HyperMesh package. There are other packages available that could have been used for this step. The Nastran model consisted of linear and quadratic, hexahedral, pentahedral and tetrahedral solid elements. Linear and quadratic, quadrilateral and triangular shell elements, and rigid elements were also used.
Figure 3.1: A cut-away view of an automotive rear-axle assembly.
Figure 3.2: A schematic drawing of the automotive rear-axle assembly.

Figure 3.3: The housing finite element mesh.
Figure 3.4: The housing menu.

Figure 14.56 shows the housing menu for the NASTRAN_EXTERNALFE housing TYPE. We constrain the housing reference frame degrees of freedom by setting UXCONSTRAINT, UYCONSTRAINT, UZCONSTRAINT, THETAXCONSTRAINT, and THETAYCONSTRAINT to TRUE. The name of the Nastran file is entered into the FILENAME field as rearaxle_housing.bdf.

NODETOLERANCE sets the tolerance used at the user defined race surface to find connecting nodes. MAXIMUMJOINTANGLE sets the maximum angle used to smooth joining shell elements. Joining elements forming angles greater than this amount are left as sharp edges. We use the recommended angle of 15 degrees in this example. For the sake of convenience, our system’s reference frame was made to coincide with the reference frame of the housing model. To do so, we set XSHIFT = YSHIFT = 0, and ZSHIFT = 44.016.

The RACE menu is shown in Figure 3.6. X, Y, and Z are the locations of the race origins in the reference frame of the CAD housing. In this case, housing reference frame is aligned with our global model reference frame, so the race coordinates are identical to the bearing coordinates. AX, AY, and AZ are the unit vectors that define the orientation of the positive race axis. AXPOSN1 and AXPOSN2 set the race length by defining the axial position of each edge from the race origin. The RACE menu inputs used for this example can be obtained from Figure 3.5.

### 3.2 The Carrier Rotor

The first rotor in the model is the carrier rotor, shown in Figure 3.7. It consists of a shaft, a carrier with four straight bevel pinions, and a hypoid gear.

Figure 3.8 shows the data required to set up the carrier rotor (rotor 1). Again, we choose to make the origin of the rotor the same as the system origin so XPOSN, YPOSN and ZPOSN are all zero. The rotor axis of rotation is parallel to the system Z axis, so AX=0, AY=0 and AZ=1.0.

We would like to apply a zero external torque on the carrier, and we would like the angular velocity of the carrier to be determined by Transmission3D. In order for this to happen, we have set TYPE to IDLER.

Since there are no shafts on this rotor, ENABLESHAFTS is left unchecked. ENABLEHYPOIDS and ENABLECARRIERS are checked, with NHYPOIDS=1 and NCARRIERS=1.

The UXCONSTRAINT, UYCONSTRAINT, UZCONSTRAINT, THETAXCONSTRAINT and THETAYCONSTRAINT flags are left unchecked because all the necessary constraints for this rotor are supplied by the roller type connectors that the carrier rests on. No additional external loads are applied, so FX, FY, FZ, MX and MY are all zero.
Figure 3.5: Housing race locations and dimensions.

Hypoid Offset = 43.983

Dimensions:
- R = 32.068, z = [-971.615:955.815]
- R = 72.5, z = [-125.0:91.0]
- R = 65.0, z = [111.336:150.812]
- R = 32.068, z = [1086.815:102.615]
- R = 36.0, x = [-114.917:95.041]
- R = 59.0875, x = [-278.046:261.986]
- R = 92.5, x = [-235.5:204.46]
Figure 3.6: Housing race menu for race 1.
Figure 3.7: Rotor 1, the carrier.
Figure 3.8: The rotor menu for the carrier rotor.
3.2.1 The Carrier

This rotor has a very complicated carrier. The carrier was first created using the CAD package Pro-E. The HyperMesh package was then used to generate NASTRAN model from this CAD model. The finite element model of the carrier is shown in Figure 3.9.

The carrier finite element mesh is primarily made up of linear tetrahedral elements. Some rigid elements have also been used. Three nodes in this finite element model are constrained. This makes sure that the carrier reference frame is attached to the mesh, and keeps its stiffness matrix non-singular.

The Nastran carrier is importing using the carrier TYPE=FECARRIER_NASTRAN within the carrier menu shown in Figure 3.11. The carrier components were all exported as a single file, so we set NFECARRIERFILES=1. The FILE menu is used to enter the file details. The carrier is shifted along the positive rotational axis by AXIALSHIFT=89.016 as shown in Figure 3.10. The carrier is a conventional carrier containing 4 bevel pinions so ENABLEPINIONS is turned ON. We enter NPINIONS=1 and NGROUPS=4 to create 4 identical copies of 1 pinion.

Since the cross member and the carrier are two separate components within the Nastran mesh file, we must define a race surface where each of the 4 'posts' of the cross-member connect to the carrier. To do so, we check the ENABLEINTERNALRACES box and set NINTERNALRACES=4. The INTERNALRACE menu is then used to enter the race parameters provided in Table 14.1.

The carrier in this example requires six races. Four of the six are required to connect the roller bearings to the carrier. The remaining two are required to connect the hypoid gear and the the rotor shaft, respectively. The bearing race inputs of Table 3.2 are entered into the RACE menu. Figure 3.12 shows the race menu for the conical race.

Table 3.2 shows the race menu inputs for the cylindrical races. The positive rotational axis is set to AX=0, AY=0, AZ=1 for each cylindrical race.
Figure 3.10: The carrier on rotor 1. $zc$ is the $z$ coordinate measured with respect to the carrier origin, and $z$ is the $z$ coordinate measured with respect to the ground.

Table 3.1: FE Carrier Internal Race Menu Input Data

<table>
<thead>
<tr>
<th>Race</th>
<th>XPOS</th>
<th>YPOS</th>
<th>ZPOS</th>
<th>DIA</th>
<th>AXPOSN1</th>
<th>AXPOSN2</th>
<th>CIRC ORDER</th>
<th>AXIAL ORDER</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>82.00</td>
<td>0.00</td>
<td>-24.72</td>
<td>25.934</td>
<td>-12.00</td>
<td>12.00</td>
<td>1</td>
<td>1</td>
</tr>
<tr>
<td>2</td>
<td>0.00</td>
<td>82.00</td>
<td>-24.72</td>
<td>229.00</td>
<td>13.00</td>
<td>15.00</td>
<td>1</td>
<td>1</td>
</tr>
<tr>
<td>3</td>
<td>-82.00</td>
<td>0.00</td>
<td>-24.72</td>
<td>229.00</td>
<td>13.00</td>
<td>15.00</td>
<td>1</td>
<td>1</td>
</tr>
<tr>
<td>4</td>
<td>0.00</td>
<td>-82.00</td>
<td>-24.72</td>
<td>229.00</td>
<td>13.00</td>
<td>15.00</td>
<td>1</td>
<td>1</td>
</tr>
</tbody>
</table>
Figure 3.11: The carrier menu.

Table 3.2: FE Carrier Cylindrical Bearing Race Input Data

<table>
<thead>
<tr>
<th>Race</th>
<th>DIA</th>
<th>AXPOSN1</th>
<th>AXPOSN2</th>
<th>CIRC ORDER</th>
<th>AXIAL ORDER</th>
</tr>
</thead>
<tbody>
<tr>
<td>2</td>
<td>62.614</td>
<td>55.150</td>
<td>101.650</td>
<td>4</td>
<td>2</td>
</tr>
<tr>
<td>3</td>
<td>62.000</td>
<td>-168.113</td>
<td>-104.613</td>
<td>4</td>
<td>2</td>
</tr>
<tr>
<td>4</td>
<td>83.578</td>
<td>68.000</td>
<td>104.000</td>
<td>1</td>
<td>1</td>
</tr>
<tr>
<td>5</td>
<td>89.000</td>
<td>-183.000</td>
<td>-123.000</td>
<td>1</td>
<td>1</td>
</tr>
</tbody>
</table>
3.2.1.1 The Ring-Carrier Rotor Shaft  

The rotor 1 shaft is modeled for the sole purpose of constraining the FE model to the rotor reference frame. The shaft begins at an AXIALPOSNSHAFT = -76.937. The inner diametral surface of the shaft connects to one of the 6 carrier races, while the nodes on the cylindrical outside surface are tied to the rotor reference frame by selecting ODCONSTRAINED=TRUE and ODTYPE=FLEXIBLE in the shaft SEGMENT menu (Figure 3.13).

3.2.1.2 The Straight Bevel Pinion  

Under the pinion sub-menu (Figure 3.14) of the carrier menu, we have set NPINIONS=1. This merely means that there is only one kind of pinion on the carrier, even though there are 4 identical copies (NGROUPS=4) of this pinion. The pinion is of TYPE=BEVEL.

The location of the crossing point of the pinion measured from the carrier origin is AXPOSN=-24.72, as shown in Figure 3.10. Note that this value is measured from the carrier origin, not the rotor origin. This ensures that if we move the carrier, the pinions move with it. PHIPOSN = 90 Degrees places the pinions at the correct azimuthal position with respect to the carrier axis.

Figure 3.15 shows the pinion tooth details, and Figure 3.16 shows the pinion tooth menu into which the data is entered. The rim for this pinion (Figure 3.17) is of TYPE=SHERICAL and consists of only one segment.

A washer is modeled by checking ENABLEWASHER in the pinion menu. The washer (Figure 3.18) limits the axial motion of the pinion along its rotational axis and defines a contact surface at the washer-pinion interface. The washer is modeled in the PINION menu under the WASHER sub-menu. The required inputs of the WASHER menu are given in Figure 3.18.

Figure 3.19 shows the inputs of the CARRIERHOLES menu. This menu defines the connecting surfaces between the cross member and carrier shown in Figure 3.10.

3.2.1.3 The Hypoid Ring Gear  

The hypoid gear mounted on the carrier is shown in Figure 3.20. Figure 3.21 shows the menu into which its data is entered. The hypoid gear tooth mesh was generated in the HypoidFaceMilled package. This model was saved in the file gear_lp37.msh. The number of teeth on this hypoid gear is NTEETH=36. The crossing point of this hypoid gear is coincident with the rotor origin, as shown in Figure 3.20. The RABASE, ZABASE, RBBASE and ZBBASE specify the location of the base surface of the tooth finite element model. Because the rotor axis points from the toe of the hypoid tooth to its heel, the AXISDIRECTION is set to SAME (The rotor axis points in the same direction as the hypoid gear axis).

The rim model of the hypoid gear consists of four segments, as shown in Figure 3.22. Segment 1 of the rim model connects with the tooth model, and segment 4 of the rim model connects with the flat annular surface of the carrier. A
Figure 3.13: The rotor 1 shaft SEGMENT menu.
Figure 3.14: The pinion menu.

Figure 3.15: The straight bevel differential pinion.
Figure 3.16: The pinion tooth menu.
Figure 3.17: The rim of the straight bevel differential pinion.

Figure 3.18: The bevel pinion washer.
Figure 3.19: The carrier holes menu.

Figure 3.20: The hypoid ring gear on rotor 1.
Figure 3.21: The hypoid menu for the carrier’s hypoid gear.

Figure 3.22: The rim of the hypoid ring gear on rotor 1.
3.3 The Propeller Shaft Rotor

The propeller shaft (Figure 3.23 forms the second rotor. This rotor consists of one shaft and one hypoid pinion resting on three four connectors.

Figure 3.24 shows the rotor menu populated with the data needed to set this rotor up. For convenience, we place the origin of this rotor at the crossing point of its hypoid pinion, at (X,Y,Z) = (0, -43.983, 0). Here 43.983 is the hypoid offset. Accordingly we have XPOSN= 0, YPOSN= -43.983 and ZPOSN= 0. The rotor axis points along the negative X axis. Hence AX=-1, AY=0 and AZ=0.

We would like to specify the speed on the propshaft. So we set its TYPE to INPUT. The speed of RPM = -6,000 is set following the right hand rule (about the rotor axis) for the sign convention. We have checked the ENABLESHAFTS and ENABLEHYPOIDS flags and set NSHAFTS to 1 and NHYPOIDS to 1. All the necessary constraints for this rotor come from the connectors that support the rotor. So we leave the UXCONSTRAINT, UYCONSTRAINT, UZCONSTRAINT, THETAXCONSTRAINT and THETAYCONSTRAINT flags unchecked. FX, FY, FZ, MX and MY are all zero.

3.3.0.1 The Hypoid Pinion  The hypoid pinion for the prop-shaft is shown in Figure 3.25. Figure 3.26 shows the menu with the pinion data. The hypoid pinion tooth finite element mesh was also generated using the Hypoid-FaceMilled package. The model was saved in file pinion lp37.msh after generation. The number of teeth NTEETH is 7. Since we chose the rotor origin to coincide with the pinion’s crossing point, we set AXIALPOSN=0. The RABASE, ZABASE, RBBASE and ZBBASE defines the base surface of the tooth finite element model are shown in Figure 3.25. Because the rotor axis points from the toe of the hypoid tooth to its heel, the AXISDIRECTION is set to SAME (The rotor axis points in the same direction as the hypoid pinion axis).

The rim model of the pinion consists of just one segment (Figure 3.27). Interpolation orders of CIRCORDER=8 and RADIALORDER=2 are used at the pinion tooth-rim interface, and at the pinion rim-shaft interface.

3.3.0.2 The Shaft  As shown in Figure 3.23, the shaft has 9 segments. The outer diameter of segments 1, 4 and 6 form races. These races forms the connection between the propshaft and the housing via roller type connectors.

The race on segment 3 connects the shaft model to the pinion rim.

The outer diameter of segment 8 is constrained. This constraint ensures that the rotor reference frame is attached to the its finite element models, and that the finite element models of the rotor have a non-singular stiffness matrix.
Figure 3.24: The rotor menu for the prop-shaft rotor.
Figure 3.25: The hypoid pinion tooth.

Figure 3.26: The hypoid pinion menu.


3.4 The Half-Shafts

The left and right wheel half shafts form rotors 3 and 4. These are shown in Figures 14.37 and 14.38. Figures 14.35 and 14.36 show the menus for these two rotors. These two rotors each consist of one straight bevel pinion and one shaft.

The origins of both the rotors are located, for convenience, at the crossing point of the straight bevel pinion and gear axes. This is at \((X, Y, Z) = (0, 0, 64.296)\). So we have \(X\text{POSN}=0, Y\text{POSN}=0\) and \(Z\text{POSN}=64.296\) for both the rotors. The left wheel half shaft (rotor 3) has its axis pointing along the +Z direction, so it has \(AX=0, AY=0\) and \(AZ=1.0\). For the right wheel half shaft (rotor 4), the axis points along the -Z direction, so it has \(AZ=-1.0\).

Since we would like to specify the angular velocity for the left half and torque on the right half shaft, their \(\text{TYPE}\) is set to \(\text{INPUT}\) and \(\text{OUTPUT}\) respectively. We specify \(\text{RPM}=-1000\) for the left half shaft and \(\text{TORQUE} = 2.5\text{E6} \) Nmm for the right half shaft. Note that since the rotor axes for the two half shafts point in opposite directions, this means that they are actually rotating in opposite directions. In order to make them spin in the same direction, sign of the RPM value for the left half is negative. We have checked the \(\text{ENABLESHAFTS}\) and \(\text{ENABLEBEVELS}\) flags and set \(\text{NSHAFTS}\) to 1 and \(\text{NBEVELS}\) to 1.

We would like to hold the wheel end of the half shafts, so we check the \(\text{UXCONSTRAINT}, \text{UYCONSTRAINT}, \text{UZCONSTRAINT}, \text{THETAXCONSTRAINT}\) and \(\text{THETAYCONSTRAINT}\) flags. \(\text{UX}, \text{UY}, \text{UZ}, \text{THETAX}\) and \(\text{THETAY}\) are all zero.

3.4.0.1 The Bevel Gear

The bevels gears on the two half shafts are identical, and are shown in Figure 14.40. Figures 14.41 and 14.42 show the menus into which the bevel gear details are entered. The rim for this bevel gear consists of only one segment (Figure 3.35).

3.4.0.2 The Shaft

The shafts models for rotors 3 and 4 are very similar. They differ only in the lengths of a few segments. The two shafts have six segments. The straight bevel gear connects to the outer diameter of segment 1. The outer diameters of segments 3 and 5 form races where the rotors connect with the housing through connectors. The
Figure 3.28: Rotor 3, the left wheel half-shaft.
Figure 3.29: Rotor 4, the right wheel half-shaft.
Figure 3.30: The rotor menu for the left wheel rotor.
Figure 3.31: The rotor menu for the right wheel rotor.
Figure 3.32: The bevel gear of rotors 3 and 4.
Figure 3.33: The menu for a bevel gear of rotors 3 and 4.

<table>
<thead>
<tr>
<th>MultiX Edit Rotor Bevel Tooth</th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>EXIT</td>
<td></td>
<td></td>
</tr>
<tr>
<td>QUIT</td>
<td></td>
<td></td>
</tr>
<tr>
<td>TOOTH</td>
<td></td>
<td></td>
</tr>
<tr>
<td>RIM</td>
<td></td>
<td></td>
</tr>
<tr>
<td>NBEVELES</td>
<td>1</td>
<td></td>
</tr>
<tr>
<td>BEVEL</td>
<td>1</td>
<td></td>
</tr>
<tr>
<td>AXIALPOSN</td>
<td>0</td>
<td></td>
</tr>
<tr>
<td>AXISDIRECTION</td>
<td>SAME</td>
<td></td>
</tr>
<tr>
<td>CIRCORDER</td>
<td>8</td>
<td></td>
</tr>
<tr>
<td>RADIALORDER</td>
<td>2</td>
<td></td>
</tr>
</tbody>
</table>

Figure 3.34: The menu for a bevel gear of rotors 3 and 4.
There are ten connectors in the model. Tables 3.3 and 13.3 show the data associated with the connectors.

Connectors 1 and 2 connect the wheel half-shafts (rotors 3 and 4, respectively) with the housing. Connectors 3 and 4 connect the carrier rotor (rotor 1) with the housing. Connectors 5, 6 and 7 connect the prop-shaft (rotor 2) to the housing. Four instances of connector 8 connect each of the four straight bevel pinions to the carrier. Finally, connectors 9 and 10 connect the carrier to the left and right half shafts.

The following sub-sections present a few of the options available for modeling connectors in Transmission3D. The bevel pinion connector is modeled with a journal bearing in the next section. The following two sections will describe how to model the remaining bearings using stiffness bearings and roller element bearings.

### 3.5.1 Journal Bearing

The bevel pinion bearing connects the four bevel pinions to the carrier cross member. This connector is modeled as a journal bearing in order to appropriately model the contact that occurs in the physical system. The journal bearing, when selected, automatically defines contact conditions on the inner and outer surfaces. This allows us to model the cross member-bearing and bearing-pinion contact. Figure 3.36 provides the inputs for the bevel pinion bearing. Note the position of a ‘pinion-to-carrier’ bearing is entered as an AXIALPOS relative to the carrier origin, and in the direction of the pinion rotational axis.
Table 3.3: Connector Locations and Orientations

<table>
<thead>
<tr>
<th>Connector</th>
<th>Name</th>
<th>X</th>
<th>Y</th>
<th>Z</th>
<th>A_x</th>
<th>A_y</th>
<th>A_z</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>HOUSING_WHEEL1</td>
<td>0</td>
<td>0</td>
<td>-918.984</td>
<td>0</td>
<td>0</td>
<td>1</td>
</tr>
<tr>
<td>2</td>
<td>HOUSING_WHEEL2</td>
<td>0</td>
<td>0</td>
<td>1138.016</td>
<td>0</td>
<td>0</td>
<td>1</td>
</tr>
<tr>
<td>3</td>
<td>CARRIERBRG1</td>
<td>0</td>
<td>0</td>
<td>-47.66354</td>
<td>0</td>
<td>0</td>
<td>1</td>
</tr>
<tr>
<td>4</td>
<td>CARRIERBRG2</td>
<td>0</td>
<td>0</td>
<td>161.67064</td>
<td>0</td>
<td>0</td>
<td>1</td>
</tr>
<tr>
<td>5</td>
<td>PINIONCYLBRG</td>
<td>-105</td>
<td>-43.983</td>
<td>0</td>
<td>-1</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>6</td>
<td>PINIONTAILBRG</td>
<td>-287.9858</td>
<td>-43.983</td>
<td>0</td>
<td>-1</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>7</td>
<td>PINIONHEADBRG</td>
<td>-200.5326</td>
<td>-43.983</td>
<td>0</td>
<td>-1</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>8</td>
<td>BEVELPINIONBRG</td>
<td>N/a</td>
<td>N/a</td>
<td>N/a</td>
<td>N/a</td>
<td>N/a</td>
<td>N/a</td>
</tr>
<tr>
<td>9</td>
<td>CARRIER_LEFTWHEEL</td>
<td>0</td>
<td>0</td>
<td>167.016</td>
<td>0</td>
<td>0</td>
<td>1</td>
</tr>
<tr>
<td>10</td>
<td>CARRIER_RIGHTWHEEL</td>
<td>0</td>
<td>0</td>
<td>-47.097</td>
<td>0</td>
<td>0</td>
<td>1</td>
</tr>
</tbody>
</table>

Figure 3.36: Connector menu for journal bearing.
### Table 3.4: Connector Stiffnesses

<table>
<thead>
<tr>
<th>Connector</th>
<th>Name</th>
<th>$K_r$ N/mm</th>
<th>$K_z$ N/mm</th>
<th>$K_{\theta_r}$ Nmm/Rad</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>HOUSING_WHEEL1</td>
<td>100000</td>
<td>100000</td>
<td>0</td>
</tr>
<tr>
<td>2</td>
<td>HOUSING_WHEEL2</td>
<td>100000</td>
<td>100000</td>
<td>0</td>
</tr>
<tr>
<td>3</td>
<td>CARRIERBRG1</td>
<td>100000</td>
<td>100000</td>
<td>1.00E+09</td>
</tr>
<tr>
<td>4</td>
<td>CARRIERBRG2</td>
<td>100000</td>
<td>100000</td>
<td>1.00E+09</td>
</tr>
<tr>
<td>5</td>
<td>PINIONCYLBRG</td>
<td>100000</td>
<td>0</td>
<td>1.00E+09</td>
</tr>
<tr>
<td>6</td>
<td>PINIONTAILBRG</td>
<td>100000</td>
<td>100000</td>
<td>1.00E+09</td>
</tr>
<tr>
<td>7</td>
<td>PINIONHEADBRG</td>
<td>400000</td>
<td>400000</td>
<td>4.00E+09</td>
</tr>
<tr>
<td>9</td>
<td>CARRIER_LEFTWHEEL</td>
<td>100000</td>
<td>100000</td>
<td>1.00E+09</td>
</tr>
<tr>
<td>10</td>
<td>CARRIER_RIGHTWHEEL</td>
<td>100000</td>
<td>100000</td>
<td>1.00E+09</td>
</tr>
</tbody>
</table>

#### 3.5.2 Stiffness Bearing

We first model the remaining connectors with stiffness bearings. The following section will discuss how to model some of these connectors with rolling element bearings. The input for the stiffness bearing HOUSING_WHEEL1 is shown in the Figure 3.37. This bearing connects the right half shaft (MEMEBER1TYPE=ROTOR) to the housing (MEMEBER2TYPE=HOUSING). The right wheel being the fourth rotor in the model, IDROTOR1 = 4 and the ID for the housing, IDHOUSING2 = 1. The bearing origin is defined at the center of the race with XPOS=0, YPOS=0, ZPOS=−918.984. The bearing axis is chosen along the positive Z axis with AX=0, AY=0 and AZ=1. The axial extent of the race form the bearing origin, AXPOSNRACE1, AXPOSNRACE2=−8 and AXPOSNRACE1, AXPOSNRACE2=8, specifies the length of the bearing. The stiffness values in the r,z and theta direction are given in the Table 13.3. Figure 3.38 shows the input menu for a stiffness bearing. The input parameters for the stiffness bearings are summarized in Table 3.3 and Table 13.3.
3.5.3 Roller Bearing

The PINIONCYCBRG is now modeled with a cylindrical roller bearing. The roller bearing interfaces the housing to the propshaft and the carrier. The inputs for connector 5 are shown in Figure 3.39. The GEOMETRY menu for the bearing is shown in Figure 3.40.

Connectors 3, 4, 6 and 7 will now be modeled using tapered roller type connecters. The following changes are available in the session file RearAxleRollerBearings.ses, which is also located in the subdirectory SAMPLES/RearAxle. For the tapered roller bearings, the TYPE menu in the GEOMETRY option is set to TAPERED and THRUSTCENTER for the bearing origin under the LOCATION_TYPE menu, shown in Figure 3.41. The schematic representation of the inputs for the PINIONHEADBRG is shown in Figure 3.42. All the tapered bearings are applied with an initial preload of AXIALCLEARANCE = -0.01 mm.

The GEOMETRY inputs for all roller bearings are summarized in Table 14.3. The CAGE properties of the bearing are auto computed by Transmission3D. The contact grid properties of the rollers are set to NPROFDIVS=0,NFACEDIVS=3,SEPTOL =0.1 and DSPROF=0.2.
Figure 3.39: Cylindrical bearing on propshaft rotor

Table 3.6: Contact Grid Parameters

<table>
<thead>
<tr>
<th>PAIR</th>
<th>ADAPTIVEGRID</th>
<th>NFACEDIVS</th>
<th>NPROFDIVS</th>
<th>SEPTOL</th>
</tr>
</thead>
<tbody>
<tr>
<td>HYPOID_HYPOID</td>
<td>ON</td>
<td>50</td>
<td>5</td>
<td>0.1</td>
</tr>
<tr>
<td>BEVEL_PINION PINION&amp;LEFTWHEEL</td>
<td>ON</td>
<td>3</td>
<td>3</td>
<td>0.1</td>
</tr>
<tr>
<td>BEVEL_PINION PINION&amp;RIGHTWHEEL</td>
<td>ON</td>
<td>3</td>
<td>3</td>
<td>0.1</td>
</tr>
</tbody>
</table>

3.6 Contact Surface Pairs

There are three contact pairs in this model. Pair 1 enforces contact between the hypoid gear (hypoid 1 of rotor 1) and the hypoid pinion (hypoid 1 of rotor 2). The second contact surface pair enforces contact between the left half shaft bevel gear (bevel 1 of rotor 3) and each of the four straight bevel pinions (pinion 1 of carrier 1 of rotor 1). Finally, the third contact pair enforces contact between the right half shaft bevel gear (bevel 1 of rotor 4) and each of the four straight bevel pinions (pinion 1 of carrier 1 of rotor 1). The menu for the hypoid pinion-gear pair is shown in Figure 3.43. The contact grid properties are defined for each contact pair separately as given in Table 3.6. Note when ADAPTIVEGRID is checked, DSPROF (contact grid point spacing in the profile direction) is set automatically.

3.7 The Analysis Setup

Figure 14.59 shows the analysis setup menu for analyzing a single time step of the system. Only one time step is being analyzed, so NTIMESTEPS is 1. The results of the file are stored in the file with name POSTFILE-
Figure 3.40: The GEOMETRY menu for Cylindrical Bearing

Figure 3.41: The GEOMETRY menu for PINIONHEADBRG Connector
Figure 3.42: Inputs of PINIONHEAD bearing

Figure 3.43: Hypoid pinion-gear pair menu.
NAME=postproc1ts.dat. Note, to run the analysis with the roller bearings, the abovementioned file name is POSTFILENAME=postproc_RollerBearings1ts.dat.

3.8 Results
The analysis of this system on a 1700 MHz Intel Pentium took eight hours to formulate and decompose all stiffness matrices. It took a further 3 hours for each time step analyzed. The following two sub-sections contain stress contours and contact patterns.

### 3.8.1 Model Results with Stiffness Bearings

Figures 3.45 through 3.50 show stress contour and contact pattern results for the analysis run using stiffness type connectors.
Figure 3.46: Contact Pattern on the hypoid pinion.
Figure 3.47: Stress contours on the hypoid gear.
Figure 3.48: Stress contours on one of the straight bevel pinions.
Figure 3.49: Stress contours on the right half shaft straight bevel gear.
Figure 3.50: Stress contours on the housing.
Figure 3.51: Stress contours on the pinion shaft with roller bearings.

3.8.2 Model Results with Roller Bearings

Figures 14.60 through 14.63 show stress contour and contact pattern results for the analysis run using the roller element bearings for connectors 3 through 7.
Figure 3.52: Stress contours on the carrier with roller bearings.
Figure 3.53: Stress contours on the housing with roller bearings.
Figure 3.54: Stress contours on the housing with roller bearings.
Table 3.7: E, P, G, α Deflection Values

<table>
<thead>
<tr>
<th>Time</th>
<th>E (mm)</th>
<th>P (mm)</th>
<th>G (mm)</th>
<th>α(rad)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.429E-004</td>
<td>-2.087E-001</td>
<td>1.755E-002</td>
<td>1.343E-001</td>
<td>1.460E-004</td>
</tr>
<tr>
<td>2.857E-004</td>
<td>-2.076E-001</td>
<td>2.059E-002</td>
<td>1.358E-001</td>
<td>1.616E-004</td>
</tr>
<tr>
<td>4.286E-004</td>
<td>-2.047E-001</td>
<td>1.030E-002</td>
<td>1.326E-001</td>
<td>1.899E-004</td>
</tr>
<tr>
<td>5.714E-004</td>
<td>-2.075E-001</td>
<td>6.870E-003</td>
<td>1.320E-001</td>
<td>1.938E-004</td>
</tr>
<tr>
<td>7.143E-004</td>
<td>-2.154E-001</td>
<td>4.805E-003</td>
<td>1.313E-001</td>
<td>1.909E-004</td>
</tr>
<tr>
<td>8.572E-004</td>
<td>-2.223E-001</td>
<td>-1.320E-004</td>
<td>1.314E-001</td>
<td>1.886E-004</td>
</tr>
<tr>
<td>1.000E-003</td>
<td>-2.337E-001</td>
<td>2.522E-002</td>
<td>1.360E-001</td>
<td>1.458E-004</td>
</tr>
<tr>
<td>1.143E-003</td>
<td>-2.170E-001</td>
<td>2.663E-002</td>
<td>1.320E-001</td>
<td>1.241E-004</td>
</tr>
<tr>
<td>1.286E-003</td>
<td>-2.187E-001</td>
<td>3.268E-002</td>
<td>1.326E-001</td>
<td>1.295E-004</td>
</tr>
<tr>
<td>1.429E-003</td>
<td>-2.150E-001</td>
<td>2.123E-002</td>
<td>1.302E-001</td>
<td>1.609E-004</td>
</tr>
<tr>
<td>1.571E-003</td>
<td>-2.097E-001</td>
<td>9.195E-003</td>
<td>1.270E-001</td>
<td>1.981E-004</td>
</tr>
</tbody>
</table>

3.8.3 EPGAlpha Results and Explanation

The E, P, G, α (V, H, G, Δσ) deflection values are generated by Transmission3D upon completion of the analysis. The values are located in a file called EPGALPHA.DAT, which can be found in the calyxtmp folder in the working directory. The file is organized left-to-right in the following order: time, E, P, G, α and top-to-bottom by the time steps for which the analysis is run. These deflection values can be used in the Gleason-CageWin and Klingelnberg Kimos design software for the purpose of re-designing the hypoid gear pair. Table 14.7 displays the E, P, G, α(rad) results for the Rear Axle model.

The E, P, G, α values are calculated by first measuring the 3-dimensional deflection at the nodes on the base surface of the loaded teeth. This is done for both the pinion and gear, separately. Next, a least squares regression of the 6 rigid body motion components is calculated to best fit the measured deflection (12 components total - 6 gear, 6 pinion). The 12 rigid body components are then converted into 6 relative motion components. Since rotation of the pinion and gear about each of their respective axis does not contribute to the deflection, we ignore these components. The remaining 4 components are the E, P, G, and α values. The sign convention for E, P, G, and α for both left and right-handed gears are described in Figures 14.64 through 14.67.
Figure 3.55: The E,P,G,Alpha sign convention for a left-handed gear.

Figure 3.56: The E,P,G,Alpha sign convention for a left-handed gear.
Figure 3.57: The E,P,G,Alpha sign convention for a right-handed gear.

Figure 3.58: The E,P,G,Alpha sign convention for a right-handed gear.
The wind turbine gear box demonstrated here consists of one planetary speed increaser stage followed by two helical stages. The ring gear of the planetary first stage is integral with the housing. The planetary pinion bearings have been modeled in detail with rolling element bearings. The model also features a spline connection between the output of the planetary stage and the input of the subsequent helical stage.

The inputs for this example can be loaded from the session file called WindTurbineGearbox.ses in the subdirectory SAMPLES/WindTurbineGearbox under the default working directory. The housing and carrier finite element mesh are also in this directory.

For this example, we have selected the Newton as the unit of force and millimeter as the unit of length. Only one speed needs to be specified to define the kinematics of this gearbox. As always, we prefer to specify the speed at the high speed end, and specify the torque at the low speed side.
Figure 4.1: The wind turbine gearbox assembly.
4.1 The Housing Model

The housing model consists of two separate parts as shown in Figure 4.2. The two parts are joined by the Ring Gear Rotor, discussed in Section 4.2. The housing is imported into Transmission3D as an FE model created using the HyperMesh FEA software package. Both housing components are saved as a single Nastran bulk data file (.bdf), with the Young’s modulous, Poisson’s ratio, and density defined within HyperMesh. To import the housing, we enter HousingBothParts.bdf as the FILENAME and define the 13 race surfaces on the housing. The RACE menu inputs are shown in Figures 4.3 and 4.4. The constraint boxes are left unselected since the housing is fully constrained by rubber bushings modeled using stiffness bearings.
Figure 4.3: Races on Housing - 1
Figure 4.4: Races on Housing - 2
4.2 The Ring Rotor

The ring rotor (Figure 4.5) consists of the planetary ring gear and one shaft. The rotor origin coincides with the global origin at XPOSN = YPOSN = ZPOSN = 0 and the positive z-axis is aligned with the global z-axis (AX = AY = 0, AZ =1). The rotor TYPE is set to ATTACHEDTOHOUSING in order to fix the ring rotor reference frame to the housing reference frame.

The ring rotor shaft is a single segment, cylindrical shaft. The shaft is modeled using the material properties of steel: YOUNGSMOD = 206000, POISSON = 0.3, and DENSITY = 7.8E-09. The same material properties are used throughout this model unless otherwise noted. The shaft begins at an AXIALPOSNSHAFT = -240.0 from the rotor origin. The ENABLEFRONTINTERFACE and ENABLEBACKINTERFACE options are used in order to connect the shaft to the conical races on the two housing components (races 9 and 10). The circular and radial order of the shaft interfaces must match the orders of the mating races on the housing for proper connection. The shaft segment is modeled within the SEGMENT menu. The segment has an INNERDIA = 1100.0, OUTERDIA = 1164.0, and LENGTH = 250.0. The inside diametral surface connects to the outer surface of the ring gear, so IDRACE is set to ON. The circular and axial orders used at this interface must match those used on the ring gear base surface. We constrain the outside of the ring shaft by enabling the ODCONSTRAINT checkbox. The AXIALORDER and CIRCORDER is identical to those used for the inner race. Since the ring rotor is attached to the housing, this shaft constrained serves as the housing constraint and there is no need for nodal housing constraints.

The ring gear is modeled within the RING menu. Here, we set AXIALPOSN = -115.0 to shift the mid-face point of the gear along the rotational axis. The involute design parameters are entered within the TOOTH menu shown in Figure 4.6. ENABLEFRONTINTERFACE and ENABLEBACKINTERFACE are selected within the tooth menu so the front and back surfaces of the ring gear also connect to housing races 9 and 10, respectively. The FRONTSHOULDERDIA and REARSHOULDERDIA are both equal to the ROOTDIA. This connects all nodes on the front and back gear tooth surfaces between the inside and root diameters to the housing.
### Figure 4.6: The ring gear tooth menu.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>NTEETH</td>
<td>80</td>
</tr>
<tr>
<td>MFACEElems</td>
<td>A</td>
</tr>
<tr>
<td>COORDORDER</td>
<td>10</td>
</tr>
<tr>
<td>DISPORDER</td>
<td>8</td>
</tr>
<tr>
<td>PROPELEVTY</td>
<td>SIMPLE</td>
</tr>
<tr>
<td>SYMMETRICTOOTH</td>
<td>TRUE</td>
</tr>
<tr>
<td>FLAINE</td>
<td>NORMAL</td>
</tr>
<tr>
<td>NORMALDAMPITCH</td>
<td>0.10000000000</td>
</tr>
<tr>
<td>NORMALPRESSANGLE</td>
<td>20.0000000000</td>
</tr>
<tr>
<td>NORMALTHICKNESS</td>
<td>8.480925890</td>
</tr>
<tr>
<td>FACEWIDTH</td>
<td>250.0000000000</td>
</tr>
<tr>
<td>HAND</td>
<td>LEFT</td>
</tr>
<tr>
<td>HELIXANGLE</td>
<td>7.4947000000</td>
</tr>
<tr>
<td>RUTERTRD</td>
<td>2.1200000000</td>
</tr>
<tr>
<td>INNERIA</td>
<td>950.7000000000</td>
</tr>
<tr>
<td>ROOTDIA</td>
<td>1047.0490000000</td>
</tr>
<tr>
<td>OUTERIA</td>
<td>1100.0000000000</td>
</tr>
<tr>
<td>DOCHAMFER</td>
<td>TRUE</td>
</tr>
<tr>
<td>ENABLEFRONDOFFSET</td>
<td>TRUE</td>
</tr>
<tr>
<td>FRONTSHOULDERIA</td>
<td>1047.0600000000</td>
</tr>
<tr>
<td>BACKSHOULDERIA</td>
<td>1047.0000000000</td>
</tr>
<tr>
<td>RADIALORDERRING</td>
<td>1</td>
</tr>
<tr>
<td>YOUNGSMOD</td>
<td>200.0000000000</td>
</tr>
<tr>
<td>POISON</td>
<td>0.5000000000</td>
</tr>
<tr>
<td>DENSITY</td>
<td>7.0000000000e+09</td>
</tr>
<tr>
<td>ALPHA</td>
<td>0.0010000000</td>
</tr>
<tr>
<td>BETA</td>
<td>1.0000000000e+07</td>
</tr>
<tr>
<td>THERMALEXPANSION</td>
<td>0.0000139003</td>
</tr>
<tr>
<td>DELTA_TEMPERATURE</td>
<td>0.0000000000e+00</td>
</tr>
<tr>
<td>TEMPLATE</td>
<td>FINESLOTTPL</td>
</tr>
</tbody>
</table>

---

**THE RING ROTOR**|

Page 125
4.3 The High Speed Pinion Rotor

The second rotor is the high speed pinion rotor. Since we would like to define the speed at this rotor, we select the rotor TYPE = INPUT, and RPM = 1809. The rotor origin is located at XPOSN = 0, YPOSN = 520.0, ZPOSN = 284 within the global reference frame, and the rotational axis is the positive z-axis of the global reference frame (AX = AY = 0, AZ = 1). The high speed pinion consists of 5 shafts and a sun gear, and its reference frame is left unconstrained in all degrees of freedom.

The pinion shaft details are shown in Figure 4.8. ENABLEFRONTINTERFACE is selected on shaft 1 in order to connect it to the back of shafts 2 and 4. Thus, ENABLEBACKINTERFACE is selected on shafts 2 and 4. ENABLEFRONTINTERFACE is also selected on shafts 2 and 5, in order to connect them to the back or shaft 3 which has ENABLEBACKINTERFACE selected. Selecting ENABLEFRONTINTERFACE on shaft 4 and ENABLEBACKINTERFACE on shaft 5 creates the races on the shafts where the sun gear will connect. The rotor’s constraint is placed on segment 6 of shaft 1.

The sun gear mid-face is positioned at an AXIALPOSN = -166.0 from the rotor origin. The involute design parameters are modeled within the TOOTH menu shown in Figure 4.9. The ENABLEBACKDOFSET and ENABLEFRONTDOFSET options are selected in order to connect the gear to shafts 4 and 5, respectively. The front and back shoulder diameters are equal to the outside diameter of the mating shafts.
Figure 4.8: The high speed pinion rotor shaft details.
Figure 4.9: The sun gear tooth menu.
4.4 The Intermediate Rotor

The intermediate rotor is shown in Figure 4.10. The rotor is positioned at $X_{\text{POSN}} = 246.31$, $Y_{\text{POSN}} = 376.611$, $Z_{\text{POSN}} = 114.0$ and its rotational axis is aligned with the positive $z$-axis of the global reference frame ($AX = AY = 0$, $AZ = 1$). We select the IDLER type for this rotor since neither a speed or torque will be defined on the rotor. The rotor contains 3 shafts and 2 sun gears and its reference frame is unconstrained in all degrees of freedom.

The intermediate rotor shaft details are shown in Figure 4.11. The first shaft is the main shaft, to which both gears connect. The second sun gear connects to the outer shaft race at segment 3 and the first gear connects to the OD race of segment 5. Segments 7 and 8 contain races on their outer diametral surfaces for connection to a roller bearing. The second and third shafts connect to the race surfaces at the outside diameter of shafts 4 and 6. The flexible rotor shaft constraint is located on the outside of segment two of the first shaft.

Shafts two and three connect to shaft 1 and to the front and back interfaces on the first sun gear. ENABLEFRONTINTERFACE is selected on shaft 2 and ENABLEBACKINTERFACE is selected on shaft 3. IDRACE is selected on the first segment of each shaft for interface with the main shaft.

The first sun gear connects to the 5th segment of shaft 1, and its mid-face is located at $AX_{\text{ALPOSN}} = 335.0$. ENABLEFRONTDOFSET and ENABLEBACKDOFSET are selected for connection to the conical races of shafts 3 and 2, respectively. The shoulder diameters of 190 are equal to the outer diameters of shafts 2 and 3. The second sun gear connects to shaft 1, segment 3 and its mid-face is located at $AX_{\text{ALPOSN}} = 170.0$. The TOOTH menus shown in Figures 4.12 and 4.13 show the involute design parameters for sun gears 1 and 2, respectively.
Figure 4.11: The intermediate rotor shaft details.
Figure 4.12: The 1st sun gear tooth menu.
Figure 4.13: The 2nd sun gear tooth menu.
4.5 The Hollow Shaft Rotor

The hollow shaft rotor is shown in Figure 4.14. The rotor contains a shaft and a sun gear. The rotor origin is located at $X\text{POSN} = 0$, $Y\text{POSN} = 0$, $Z\text{POSN} = 449.0$ and its rotational axis is aligned with the positive $z$-axis of the global reference frame. The rotor reference frame is left unconstrained in all degrees of freedom. We set the rotor $\text{TYPE} = \text{IDLER}$ since we do not want to define a torque or speed at the rotor.

The rotor shaft modeled within the SHAFT menu and the first segment begins at an $AXIAL\text{POSNSHAFT} = -300.0$ from the rotor origin. The shaft SEGMENT menu details are shown in Figure 4.15. Segment 1 contains a FLEXIBLE constraint at the inside diameter and a bearing race on its OD. The spline connects to the outer surfaces of segment 2 and the gear connects to the outside of segment 4. Bearing races are also located at the ID of segment 5 and the OD of segment 6.

The sun gear mid-face is coincident with the rotor origin ($AXIAL\text{POSN} = 0$). The involute design parameters are entered within the TOOTH menu shown in Figure 4.16.
Figure 4.15: The hollow shaft rotor shaft details.
Figure 4.16: The sun gear tooth menu.
4.6 The Planetary Sun Rotor

The fifth rotor in the model is the planetary sun rotor. The planetary sun rotor origin is located at \(X\text{POSN} = 0, Y\text{POSN} = 0, Z\text{POSN} = 102.25\) and its axis is aligned with the positive \(z\)-axis of the global reference frame. The planetary sun is also an IDLER rotor type. There are 2 shafts and a sun gear on the rotor and its reference frame is unconstrained in all degrees of freedom.

The planetary sun rotor shaft details are shown in Figure 4.18. The first shaft consists of two segments; the first segment contains the rotor shaft constraint on both its inside and outside diametral surfaces. The outside of the second shaft segment connects to the sun gear. ENABLEFRONTINTERFACE is enabled for the first shaft in order to connect it to the back of the second shaft. ENABLEBACKINTERFACE is selected for the second shaft to complete the connection of the two shafts. The external spline connects to segment 5 of the second shaft. This spline mates with the internal spline connected to the hollow rotor shaft. The sixth shaft segment contains an ODRACE for connection to a bearing.

The planetary sun gear mid-face is shifted by \(AXIAL\text{POSN} = -12.75\) along the rotational axis. The sun TOOTH menu is shown in Figure 4.19. ENABLEFRONTDOFS is selected to connect the front of the sun gear to the back of the second shaft at the same interface where the two shafts connect.
Figure 4.18: The planetary sun rotor shaft details.
Figure 4.19: The sun gear tooth menu.
4.7 The Planetary Carrier Rotor

The planetary carrier rotor is the sixth rotor in the model. The rotor is located at XPOSN = YPOSN = 0, ZPOSN = -115.0 in the global reference frame. The rotational axis of the rotor is aligned with the positive z-axis of the global reference frame. Since we would like to define a torque at this rotor, we select the TYPE = OUTPUT and TORQUE = 3.2261E+08. The rotor contains one shaft and one carrier and is left unconstrained in all degrees of freedom.

4.7.1 The Shaft

The carrier shaft details are shown in Figure 4.21. The shaft begins at an AXIALPOSN = -2351.0 from the carrier rotor origin. The shaft consists of 7 segments. Segments 1 and 3 contain outer races which connect to bearings, while segments 6 and 7 have outer races that connect directly to the carrier. The rotor’s shaft constraint is located on the inner surfaces of segments 6 and 7.

4.7.2 The Carrier

The carrier and its subcomponents are modeled within the CARRIER menu shown in Figure 4.22. The carrier in this example is modeled and meshed in an external FEA software and imported into Transmission3D using the FECARRIER_NASTRAN carrier TYPE. The imported carrier reference frame is shifted along the rotor rotational axis by AXIALSHIFT = -617.0. The carrier contains 3 identical copies of 1 pinion, so NPINIONS = 1 and NGROUPS = 4.

Within the FILE menu, we enter the information about the Nastran FE mesh file to be imported. For this example, we enter NASTRANFILENAME = Carrier.bdf, SUBTREENAME = Carrier, and PREFERREDCUTTINGDIR = Z.

The carrier has 4 race interfaces so NRACES is set to 4. Figure 4.23 shows the race locations and input parameters that are entered into the RACE menu. Races 1 and 2 connect the carrier to bearings, and races 3 and 4 connect the carrier to the carrier shaft.
Figure 4.21: The carrier rotor shaft details.

Table 4.1: The GROUP menu inputs

<table>
<thead>
<tr>
<th>Group</th>
<th>AXPOSN (mm)</th>
<th>THETA (deg)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0.00</td>
<td>120</td>
</tr>
<tr>
<td>2</td>
<td>0.00</td>
<td>240</td>
</tr>
<tr>
<td>3</td>
<td>0.00</td>
<td>360</td>
</tr>
</tbody>
</table>
Figure 4.22: The carrier menu.
WIND TURBINE GEARBOX

PINION1
RADIAL POS 308.0
ANGULAR POS -90.0

RACE 1
DIA 400.0
Z1 804.9 Z2 851.1

RACE 2
DIA 459.94
Z1 287.8 Z2 366.2

RACE 3
DIA 330.00
Z1 10.0 Z2 156.0

RACE 4
DIA 323.00
Z1 156.0 Z2 309.0

Pinion pin back side
RADIUS 70.0
Z3 734.0 Z4 780.0

Pinion pin front side
RADIUS 70.0
Z1 425.0 Z2 506.0
Figure 4.24: The carrier pinion assembly.
4.7.3 The Pinions

The carrier pin, pinion bearing, pinion shaft, and pinion inputs are all considered part of the pinion assembly shown in Figure 4.24. The pinion assembly inputs are entered within the PINION menu. The RADPOSN and THETAPOSN are defined within this menu as 308.0 and -90, respectively. These inputs are graphically illustrated in Figure 4.23. The TYPE_PIN and TYPE_PINIONSHAFT entries are both set to COMPOUND, which allows the pin and pinion shaft to be represented by a shaft with multiple segments in the same way the rotor shafts presented up to this point have been modeled. The pinion bearing is represented by two cylindrical roller bearings so NBEARINGS is set to 2.

Figure 4.23 shows the pinion pin front and back side carrier race definition parameters that are entered into the CARRIERHOLES menu. The carrier pin details are shown in Figure 4.26. The pin inputs are entered within the PIN-SHAFT submenu of the PINION menu. The pin is modeled with 3 shaft segments, with segments 1 and 3 connecting to the carrier and segment two connecting to the pinion bearing. We use AXIALORDER = CIRCORDER = 2 for both pin-carrier interfaces, as well as the pin-bearing race interface.

The pinion bearings fit between the outside of the 2nd carrier pin shaft segment and the inside of the 1st pinion shaft segment. The bearing race locations are set within the BEARING menu. Figure 4.27 shows the pinion bearings race location details. The roller input parameters are entered into the GEOMETRY menu shown in Figure 4.28. The GEOMETRY menu inputs of the two bearings are identical.

The PINIONSHAFT menu contains the inputs for the shaft that connects to the outside of the pinion bearing and the inside of the pinion base surface. We model this shaft using 2 segments; the first exists solely to provide a location for the pinion shaft constraint, while the second segment allows for the connection between bearing and pinion base. The pinion shaft details are shown in Figure 4.29.

The pinion deck inputs are entered into the DECK submenu of the PINION menu. The base surface CIRCORDER and AXIALORDER are entered into this menu and must match the orders at the outer surface of the pinion shaft. The involute tooth design parameters are entered within the TOOTH menu shown in Figure 4.30.
Figure 4.26: The carrier pin.
Figure 4.27: The carrier pinion bearings.

Figure 4.28: The carrier pinion bearing geometry menu.
Figure 4.29: The carrier pinion shaft.
Figure 4.30: The carrier pinion tooth menu.
4.8 The Bushing Rotors

Rotors 7 and 8 are the housing bushings. The left bushing is rotor 7 and its origin is located at XPOS = 750.0, YPOS = 0, ZPOS = -358. The right bushing rotor is symmetric about the z-axis with its origin at XPOS = -750, YPOS = 0, ZPOS = -358. Each rotor’s rotational axis is aligned with the global z-axis (AX = AY = 0, AZ = 1). Each rotor’s TYPE = ATTACHEDTOHOUSING so the bushings are connected to the housing reference frame. The left and right bushings connect to stiffness bearings 15 and 16, respectively. Since the stiffness bearings connect the housing to ground, we can release the housing reference frame constraints within the HOUSING menu, as was described in the housing section. Figure 4.32 shows the shaft details for the left and right bushing shafts. The outer race connects to the housing race and the inner race connects to the bearing. The left and right shaft menu inputs are identical.
Figure 4.32: The bushing shaft.
4.9 Connectors

The wind turbine example contains 18 connectors: 8 roller element bearings, 9 simplified stiffness bearings, and a spline connector. The locations of the bearings and their reference frame’s orientation are shown in Figures 4.33 through 4.36. Table 13.2 shows the race locations of each connector. Connectors 11 and 18 have coincident origin and race locations; the stiffness connector (connector 18) supports the thrust load, while the roller bearing supports the radial and bending loads. The connector TYPE and location data is entered into the CONNECTOR menu within the EDIT menu.
Connector 15:
LEFT_BUSHING
\( \{X, Y, Z\} = \{750.0, 0, -358.0\} \)
\( (AX, AY, AZ) = (0, 0, 1) \)

Connector 16:
RIGHT_BUSHING
\( \{X, Y, Z\} = \{-750.0, 0, -358.0\} \)
\( (AX, AY, AZ) = (0, 0, 1) \)

Figure 4.34: The stiffness bearing locations.
Figure 4.35: The stiffness bearing locations.
Table 4.2: Connector Race Locations

<table>
<thead>
<tr>
<th>ID</th>
<th>IROTOR</th>
<th>AXPOS1</th>
<th>AXPOS2</th>
<th>DIA</th>
<th>IROTOR2</th>
<th>AXPOS1</th>
<th>AXPOS2</th>
<th>DIA</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td>RACE1</td>
<td>RACE1</td>
<td>RACE1</td>
<td></td>
<td>RACE2</td>
<td>RACE2</td>
<td>RACE2</td>
</tr>
<tr>
<td>1</td>
<td>Rotor2</td>
<td>-29.0</td>
<td>29.0</td>
<td>100.0</td>
<td>Housing1</td>
<td>-29.0</td>
<td>29.0</td>
<td>180.0</td>
</tr>
<tr>
<td>2</td>
<td>Rotor2</td>
<td>-26.5</td>
<td>26.5</td>
<td>110.0</td>
<td>Housing1</td>
<td>-23.0</td>
<td>23.0</td>
<td>200.0</td>
</tr>
<tr>
<td>3</td>
<td>Rotor2</td>
<td>-26.5</td>
<td>26.5</td>
<td>110.0</td>
<td>Housing1</td>
<td>-23.0</td>
<td>23.0</td>
<td>200.0</td>
</tr>
<tr>
<td>4</td>
<td>Rotor3</td>
<td>-23.0</td>
<td>23.0</td>
<td>100.0</td>
<td>Housing1</td>
<td>-23.0</td>
<td>23.0</td>
<td>180.0</td>
</tr>
<tr>
<td>5</td>
<td>Rotor3</td>
<td>-19.0</td>
<td>19.0</td>
<td>160.0</td>
<td>Housing1</td>
<td>-25.0</td>
<td>25.0</td>
<td>240.0</td>
</tr>
<tr>
<td>6</td>
<td>Rotor3</td>
<td>-19.0</td>
<td>19.0</td>
<td>160.0</td>
<td>Housing1</td>
<td>-25.5</td>
<td>25.5</td>
<td>240.0</td>
</tr>
<tr>
<td>7</td>
<td>Rotor4</td>
<td>-16.5</td>
<td>16.5</td>
<td>280.0</td>
<td>Housing1</td>
<td>-16.5</td>
<td>16.5</td>
<td>350.0</td>
</tr>
<tr>
<td>8</td>
<td>Rotor4</td>
<td>-26.0</td>
<td>26.0</td>
<td>240.0</td>
<td>Housing1</td>
<td>-26.0</td>
<td>26.0</td>
<td>320.0</td>
</tr>
<tr>
<td>9</td>
<td>Rotor4</td>
<td>-26.0</td>
<td>26.0</td>
<td>240.0</td>
<td>Housing1</td>
<td>-26.0</td>
<td>26.0</td>
<td>320.0</td>
</tr>
<tr>
<td>10</td>
<td>Rotor5</td>
<td>-75.0</td>
<td>75.0</td>
<td>185.0</td>
<td>Rotor4</td>
<td>-75.0</td>
<td>75.0</td>
<td>218.0</td>
</tr>
<tr>
<td>11</td>
<td>Rotor6</td>
<td>-23.0</td>
<td>23.0</td>
<td>400.0</td>
<td>Housing1</td>
<td>-23.0</td>
<td>23.0</td>
<td>500.0</td>
</tr>
<tr>
<td>12</td>
<td>Rotor6</td>
<td>-28.0</td>
<td>28.0</td>
<td>459.94</td>
<td>Housing1</td>
<td>-28.0</td>
<td>28.0</td>
<td>580.0</td>
</tr>
<tr>
<td>13</td>
<td>Rotor6</td>
<td>-90.0</td>
<td>90.0</td>
<td>380.0</td>
<td>Ground</td>
<td>-90.0</td>
<td>90.0</td>
<td>560.0</td>
</tr>
<tr>
<td>14</td>
<td>Rotor6</td>
<td>-25.0</td>
<td>25.0</td>
<td>380.0</td>
<td>Ground</td>
<td>-25.0</td>
<td>25.0</td>
<td>500.0</td>
</tr>
<tr>
<td>15</td>
<td>Rotor7</td>
<td>-55.0</td>
<td>55.0</td>
<td>200.0</td>
<td>Ground</td>
<td>-55.0</td>
<td>55.0</td>
<td>180.0</td>
</tr>
<tr>
<td>16</td>
<td>Rotor7</td>
<td>-55.0</td>
<td>55.0</td>
<td>200.0</td>
<td>Ground</td>
<td>-55.0</td>
<td>55.0</td>
<td>180.0</td>
</tr>
<tr>
<td>17</td>
<td>Rotor5</td>
<td>-5.0</td>
<td>5.0</td>
<td>125.0</td>
<td>Rotor4</td>
<td>-5.0</td>
<td>5.0</td>
<td>150.0</td>
</tr>
<tr>
<td>18</td>
<td>Rotor6</td>
<td>-23.0</td>
<td>23.0</td>
<td>400.0</td>
<td>Housing1</td>
<td>-23.0</td>
<td>23.0</td>
<td>500.0</td>
</tr>
</tbody>
</table>
Connector 13: MAINBRG
\[ (X, Y, Z) = (0, 0, -2038.21) \]
\[ (AX, AY, AZ) = (0, 0, 1) \]

Connector 14: BLADE_LOAD
\[ (X, Y, Z) = (0, 0, -2441.0) \]
\[ (AX, AY, AZ) = (0, 0, 1) \]

Figure 4.36: The stiffness bearing locations.
4.9.1 Roller Bearings

The 8 roller bearings shown in Figure 4.33 consist of 4 cylindrical and 4 tapered roller bearings. The roller bearing TYPE is set within the GEOMETRY submenu of the CONNECTOR menu. Tables 4.3 and 4.4 list the remaining GEOMETRY menu inputs for the cylindrical and tapered type roller bearings, respectively. All of the roller bearings contain a single row of rollers, and any inputs not listed can be assumed to be zero.
4.9.2 Stiffness Bearings

Connectors 7-9 and 13-18 are stiffness TYPE bearings. Stiffness bearings are defined by a numerical stiffness value in each degree of freedom by selecting the DIAGONAL stiffness matrix type. Table 4.5 provides the stiffness values entered into the CONNECTOR menu. Connector 14 is a zero stiffness bearing located at the free end of the carrier shaft as shown in Figure 4.36. We place a load on the shaft using this dummy bearing by setting LOAD = TRUE and FX = -1E+05 within the CONNECTOR menu.

<table>
<thead>
<tr>
<th>ID</th>
<th>$k_x$</th>
<th>$k_y$</th>
<th>$k_z$</th>
<th>$k_{\theta_x}$</th>
<th>$k_{\theta_y}$</th>
<th>$k_{\theta_z}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>7</td>
<td>2.25E+06</td>
<td>1.93E+06</td>
<td>0</td>
<td>3.92E+07</td>
<td>3.28E+07</td>
<td>0</td>
</tr>
<tr>
<td>8</td>
<td>8.85E+06</td>
<td>8.86E+06</td>
<td>1.68e+06</td>
<td>1.35E+10</td>
<td>1.35E+10</td>
<td>0</td>
</tr>
<tr>
<td>9</td>
<td>9.65E+06</td>
<td>9.56E+06</td>
<td>1.82e+06</td>
<td>1.49E+10</td>
<td>1.47E+10</td>
<td>0</td>
</tr>
<tr>
<td>13</td>
<td>1.25E+06</td>
<td>1.25E+06</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>14</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>15</td>
<td>1.00E+04</td>
<td>1.00E+04</td>
<td>1.00e+03</td>
<td>1.00E+08</td>
<td>1.00E+08</td>
<td>0</td>
</tr>
<tr>
<td>16</td>
<td>1.00E+04</td>
<td>1.00E+04</td>
<td>1.00e+03</td>
<td>1.00E+08</td>
<td>1.00E+08</td>
<td>0</td>
</tr>
<tr>
<td>17</td>
<td>0</td>
<td>0</td>
<td>1.00e+08</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>18</td>
<td>0</td>
<td>0</td>
<td>1.00e+08</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
</tbody>
</table>
4.9.3 Spline Connector

Connector 10 is an EXTERNALSPLINE connector TYPE. The spline joins rotors 4 and 5 and allows torque to flow between the two rotors. Figure 4.37 shows the CONNECTOR menu inputs for the spline connector. The spline is modeled with straight sided teeth and the contact is modeled on both sides of the teeth by setting CONTACTTYPE = DOUBLESIDEDED.

Figure 4.37: The CONNECTOR menu for the spline connector.
Table 4.6: Pair Menu Inputs

<table>
<thead>
<tr>
<th>PAIR</th>
<th>IROTOR1</th>
<th>ISUN1</th>
<th>IROTOR2</th>
<th>ISUN2</th>
<th>SEPTOL</th>
<th>DPROF</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>2</td>
<td>1</td>
<td>3</td>
<td>2</td>
<td>1.0</td>
<td>0.158758</td>
</tr>
<tr>
<td>2</td>
<td>3</td>
<td>1</td>
<td>4</td>
<td>1</td>
<td>1.0</td>
<td>0.229122</td>
</tr>
<tr>
<td>3</td>
<td>5</td>
<td>1</td>
<td>Rotor6CARRIER1</td>
<td>Pinion1DECK1</td>
<td>0.1</td>
<td>0.193430</td>
</tr>
<tr>
<td>4</td>
<td>1</td>
<td>1</td>
<td>Rotor6CARRIER1</td>
<td>Pinion1DECK1</td>
<td>0.2</td>
<td>0.227922</td>
</tr>
</tbody>
</table>

4.10 Pairs

The model contains 4 gear pairs, which are defined within the EDIT > PAIRS menu. Table 4.6 shows the PAIRS menu inputs used in this model. The DPROF values was determined by the equation: $DPROF = \frac{(2 \times \text{HertzSemiWidth})}{(2 \times \text{NPROFDIVS}) - 1}$. The Hertz semi width is determined within the information menu in iGlass under the Contact tab when a central profile contact point is selected. The maximum value found for each contact pair was used to calculate the DPROF value. Alternatively, the ADAPTIVEGRID option could be turned ON. The analysis time is increased when using this option, so it may not be viable for models with a large number of contact pairs, or when using higher resolution contact grids.
CHAPTER 5

SIMPLE BALL BEARING-SHAFT MODEL

Transmission3D can be used to create a simple bearing-shaft model such as the one displayed in Figure 5.1. The advantage of this simple model is that it requires little computer processing time to obtain a detailed contact model of the bearing’s roller elements. We will demonstrate how to model the ball bearing shaft model in Transmission3D, as well as show how rotor forces can be applied to approximate the load on the bearing. The session file SimpleBallBearingShaft.ses is located in the subdirectory SAMPLES/SimpleBallBearingShaft within the default directory. The system is modeled using SI units (N,mm,s) for the menu inputs. The model requires one rotor and one connector, so we set NROUTERS=1, ENABLECONNECTORS = TRUE, and NCONNECTORS = 1 in the EDIT menu. Doing so enables the ROTOR and CONNECTOR menu buttons as shown in Figure 5.2.
Figure 5.1: The Ball-Bearing Shaft FE Model.
Figure 5.2: The edit menu.
5.1 The Input Rotor

The only rotor in the model is labeled the Input Rotor. This rotor requires just a single shaft, so we set ENABLE-SHAFTS=TRUE, NSHAFTS=1 within the ROTOR menu (Figure 5.3). The rotor origin is placed at the global origin by setting XPOS = 0, YPOS = 0, and ZPOS = 0. The direction of the rotational axis is set to be the positive Z-axis (AX = 0, AY = 0, AZ = 1). Choosing TYPE=INPUT allows us to specify the rotor’s speed. We choose the speed as RPM = 1000, which follows the right hand rule about the rotational axis of the rotor. We have the option to constrain the input rotor, but we choose not to do so since the ball bearing will fully constrain the rotor. So we leave UXCONSTRAINT, UYCONSTRAINT, UZCONSTRAINT, THETAXCONSTRAINT, AND THETAYCONSTRAINT unchecked. We also have the option to apply an external force and/or moment to the rotor. We will defer to doing so until later in this example however, so we leave FX, FY, FZ, MX, and MY equal to zero for the time being.

5.1.1 The Input Shaft

The input shaft details are shown in Figure 5.4. Within the shaft menu we set the shaft origin to be coincident with the rotor origin by setting AXIALPOSNSHAFT = 0. The material properties of the shaft are also set in the shaft menu. Here we use the default properties for steel: YOUNGSMOD = 2000, POISSON = 0.3, and DENSITY = 7.8E-09. The shaft is modeled with just one segment, so we set NSEGMENTS = 1.

The shaft segment is modeled in the segment menu. Here we define the remaining inputs for the input rotor shaft, which can be found in Figure 5.4. The inside diameter of the shaft is constrained to the rotor reference frame in order to keep the rotor stiffness matrix non-singular. A race surface is defined on the outer diameter of the shaft segment in order to mate the inner race of the ball bearing.

5.2 The Ball Bearing Connector

The ball bearing shown in Figure 5.1 is modeled within the connectors menu. The bearing origin is set to XPOS = 0, YPOS = 0, ZPOS = 50. The bearing rotational axis is colinear with the shaft rotational axis (AX = 0, AY = 0, AZ = 1).
Figure 5.4: Input Rotor Shaft

Figure 5.5: The ball bearing geometry menu.
The inner and outer race locations are set by entering \( \text{AXIALPOSN1RACE1} = \text{AXIALPOSN1RACE2} = -3.9375 \) and \( \text{AXIALPOSN2RACE1} = \text{AXIALPOSN2RACE2} = 3.9375 \). These positions are entered relative to the bearing origin, and along the bearing rotational axis. The inner and outer race diameters are \( \text{DIARACE1} = 60 \) and \( \text{DIARACE2} = 110 \), respectively. The members which connect to the inner and outer bearing races are also set in this menu. We choose \( \text{MEMBER1TYPE=} \text{ROTOR} \) and \( \text{IDROTOR1}=1 \) to connect rotor 1 to the inner race and \( \text{MEMBER2TYPE=} \text{GROUND} \) to connect the outer race to ground. Grounding the outer bearing race effectively constrains the bearing connector within the system. There is no need to apply a reaction torque in this particular example. The \( \text{REFERENCE\_RACE} \) is set to \( \text{OUTER} \).

### 5.2.1 Roller Bearing Submenus

The first roller bearing submenu is the connectors menu (Figure 5.5). The inputs that describe the specific race and roller geometry are entered into this menu. Figure 5.6 provides a visual explanation of the geometry menu inputs. In order to model a single-row ball bearing, the \( \text{TYPE} \) is set to \( \text{GENERAL} \) and \( \text{NROWS} = \text{IROWS} = 1 \). The \( \text{ROLLERCROWNCURV} \) is the curvature of the roller. This is typically the inverse of the radius of the roller element. \( \text{RACE1CROWNCURV} \) and \( \text{RACE2CROWNCURV} \) are the curvatures of the inner and outer races, respectively. The absolute value of these are typically slightly less than the roller curvature. A positive value relieves the edges of the roller.

The bearing cage is modeled in the cage submenu within the connector menu. For this example, \( \text{AUTOCOMPUTE} \) is turned ON. The contact grid menu is used to enter the contact grid parameters for the roller-race contact surfaces. These inputs are similar to the gear contact pair parameters entered into the pairs menu in previous examples. For the time being, we leave these to their default values (SEPTOL = 0.1, NPROFDIVS = 0, DSPROF = 0.02, and NFACEDIVS = 3). The FEModel menu sets the number and type of finite elements to be used for the rollers. We use \( \text{ELEMTYPE=} \text{QUADRATIC} \), \( \text{NCIRCDIVS} = 32 \), and \( \text{NCIRCFIVS2} = 16 \) in this case. The material menu sets the roller material properties, which are the same as those used for the input rotor. The runout menu sets the runout error and is not used for this particular example.
5.3 Analyzing the Ball Bearing-Shaft Model

We now demonstrate a simple analysis setup that generates detailed results for the ball bearing-shaft model. Using *Transmission3D* it is possible to apply a concentrated load or define displacement at a rotor origin. We do so within the rotor menu, shown in Figure 5.7. In this example we apply a 650 N force in both the x and y-direction. We select DISP_TYPE=GLOBALFRAME to define the direction of the force with respect to the global reference frame.

The analysis is set up within the setup and range menus shown in Figures 5.8 and 5.9. The setup menu input POSTFILENAME writes the analysis results to the specified file, `postproc.dat`. Within the range menu we set NTIMESTEPS=11 and DELTATIME=0.006. The total time, $T_{cyc} = \frac{\Delta t}{n_{timesteps} - 1}$, is the time for the shaft to go through one full rotation.
Figure 5.8: The analysis setup menu.

Figure 5.9: The range menu.
For detailed bearing contact analysis we change the connector contact grid parameters described earlier in the chapter. Figure 5.10 shows the parameters used for the analysis. A simple model affords us the option to increase the contact grid resolution without greatly affecting computation time. Adding a force to the rotor which approximates the load on the bearing allows us to obtain reasonably accurate results of the contact pressure distribution on the bearing roller and race surfaces. Figure 5.12 shows the pressure distribution on the critically loaded roller, while Figure 5.11 shows the stress contours of the system.
Figure 5.12: The contact grid on the ball bearing.
A DOUBLE-HELICAL PLANETARY SYSTEM

The Double-Helical Planetary system presented in this chapter demonstrates how to model a complex gear rim using an external FE mesh. We use a simple rigid carrier in this model. Since this model contains multiple rows of gears, we discuss how to model the planetary pinions with multiple "decks". A session file named DoubleHelicalTwoRingsJournalsNastран can be found in the subdirectory SAMPLES/DoubleHelicalTwoRingsJournals under the default working directory. We use SI units (N,mm,s) as the standard units for this example.

Figure 6.1 shows the assembled finite element model. The model contains 4 rotors and 5 connectors, so we set NROTORS = 4, ENABELECONNECTORS = TRUE, and NCONNECTORS = 5 in the EDIT menu. The model requires the use of pairs so ENABLEPAIRS = TRUE. The first rotor is labeled the Sun Rotor. The next two rotors each contain a ring gear and rim model made up of multiple shafts. We label these two rotors Ring1 Rotor and Ring2 Rotor. The final rotor is the Carrier Rotor. This rotor contains the rigid carrier, planetary pinions, and the output shaft.

We choose to define the speed at the sun rotor and torque at the carrier rotor, so we set TYPE = INPUT and TYPE = OUTPUT. The speed and torque defined on each respective rotor is RPM = -1000 and TORQUE = -1000000. The ring1 and ring2 rotors remain stationary, so we choose TYPE = INPUT and RPM = 0 for both. The global origin is located at the free end of the sun rotor as shown in Figure 6.1.

6.1 The Sun Rotor

The sun rotor (Figure 6.2) is modeled in the rotor menu by setting ROTOR = 1 (Figure 6.3). The rotor origin is coincident with the global origin so we set XPOS = YPOS = ZPOS = 0. The rotor axis is set to the positive z-axis by choosing AX = AY = 0 and AZ = 1. The rotor TYPE = OUTPUT with a TORQUE = -1000000. The negative torque value means the left hand rule is used to determine the torque direction. The rotor consists of two shafts (ENABLESHAFTS = TRUE, NSHAFTS = 2) and two sun gears (ENABLESUNS = TRUE, NSUNS = 2). The rotor is constrained by two stiffness bearings, so we need not constrain the rotor reference frame. No external loads or moments are applied to the rotor in this case.
Figure 6.1: The double-helical planetary system.

Figure 6.2: The Sun rotor.
6.1.1 The Sun Rotor Shafts

The sun rotor shafts, pictured in Figures 6.4 and 6.5, are modeled within the shaft menu. Shaft 1 consists of four segments so we set NSEGMENTS = 4. AXIALPOSNSHAFT = 0 places the shaft origin coincident with the rotor origin. The shaft contains 3 races and a constraint. The two races on the inside diameter of the shaft provide the joining surface for connectors 1 and 2. The third race is required to join the two sun gears and shaft 2 to the outer diameter surface of segment 4. The shaft constraint ensures the rotor stiffness matrix is non-singular and is placed on the outer diameter surface of segment 1.

Shaft number two is a single segment shaft (NSEGMENTS = 1) located between the two sun gears on the sun rotor. AXIALPOSNSHAFT = 73.00 for shaft 2. Choosing ENABLEFRONTINTERFACE = ENABLEBACKINTERFACE = TRUE defines a race surface on each end of the shaft. This allows us to connect the ends of the shaft to the front of sun 1 and back of sun 2 as shown in Figure 6.2. For each interface, we set the RADIALORDER = 2 and CIRCORDER = 4. The inside diameter race creates the joining surface on the second shaft for segment 4 of shaft 1.

The material type for both shafts is steel with the following material properties: Young’s modulus = 2.06e+05, Poisson’s ratio = 0.3, and density = 7.8e-09. We check UNIFORMMATERIAL in order to apply the same material properties to each shaft segment. The segment menu is used to enter the information about each specific shaft segment. These inputs are covered sufficiently within Figures 6.4 and 6.5 so we do not discuss them in great detail. The segment menu for segment 1 of shaft 1 is provided as a reference in Figure 6.6.

6.1.2 The Sun Gears

The sun gear inputs are entered into the sun menu within the rotor 1 menu. Figure 6.7 shows the first of the two gears. The first gear is located at an AXIALPOSN = 66.50 with respect to the rotor origin. Figure 6.8 shows the tooth menu inputs for sun 1. ENABLEFRONTDOFSET is checked in order to mate the front gear surface to the back of shaft number two. The gear is of HAND = RIGHT with a HELIXANGLE = 30 degrees.
Figure 6.4: The sun rotor shaft 1 details.

Figure 6.5: The sun rotor shaft 2 details.
Figure 6.6: The segment menu for segment 1 of shaft 1.

Figure 6.7: The sun gear details for sun 1.

The second sun gear has an AXIALPOSN = 83.50. Within the tooth menu of gear 2, ENABLEBACKDOFSET is checked in order to connect the back side of the gear to the front of shaft 2. The hand of the second gear is HAND = LEFT. Aside from the differences just mentioned, the inputs of Figures 6.7 and 6.8 are common to both gears.
Figure 6.8: The tooth menu for sun gear 1.
6.2 The Ring Rotors

The ring rotors each consist of a ring gear (ENABLERINGS = TRUE, NRINGS = 1), two shafts (ENABLESHAFTS = TRUE, NSHAFTS = 2), and a carrier (ENABLECARRIER = TRUE, NCARRIERS = 1). The coordinates of the origin of the rotor reference frame are set to XPOS = 0, YPOS = 0, ZPOS = 60.00 for rotor 2 and XPOS = 0, YPOS = 0, ZPOS = 75.00 for rotor 3. The rotor axis are set to the global z-axis by entering AX = 0, AY = 0, and AZ = 1 for both rotors. We would like to keep these rotors stationary so we choose TYPE = INPUT and RPM = 0. We must constrain the rotor reference frames so we check UXCONSTRAINT, UYCONSTRAINT, UZCONSTRAINT, THETAXCONSTRAINT, and THETAYCONSTRAINT. We do not apply any rotor displacement so we leave UX, UY, UZ, THETAX, and THETAY = 0. The complete ROTOR menus for rotors 2 and 3 are shown in Figures 6.9 and 6.9.

6.2.1 The Ring Gear

The ring gear details for the rotor 2 gear are provided in Figure 6.11. The rotor 3 ring gear is identical, with the exception of the HAND of the helical teeth, and the location of the interface with the rim shaft. Within the ring menu the AXIALPOSN is set to 6.50 for both gears. The gear tooth parameters are entered into the tooth menu (Figures 6.12 and 6.13). The gear-rim shaft interface is defined on the front side of the rotor 2 ring gear (ENABLEFRONTDOFSET = TRUE) and on the back of the rotor 3 gear (ENABLEBACKDOFSET = TRUE). The base surface order of interpolation is set in the base menu to CIRCORDER = 32 and AXIALORDER = 2.
Figure 6.10: The Ring2 rotor menu.
Figure 6.11: The ring gear details.

Figure 6.12: The ring gear tooth menu.
<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>TEETH</td>
<td>100</td>
</tr>
<tr>
<td>NUMBER</td>
<td>1</td>
</tr>
<tr>
<td>COORDORDER</td>
<td>10</td>
</tr>
<tr>
<td>disproorder</td>
<td>1</td>
</tr>
<tr>
<td>PROFILETYPE</td>
<td>SIMPLE</td>
</tr>
<tr>
<td>SYMMETRICTOOTH</td>
<td>TRANSVERSE</td>
</tr>
<tr>
<td>XVERSEDIMIN</td>
<td>152016990000</td>
</tr>
<tr>
<td>XVERSEPRESANGLE</td>
<td>192000000000</td>
</tr>
<tr>
<td>XVERSETHICKNESS</td>
<td>171599000000</td>
</tr>
<tr>
<td>FACEWIDTH</td>
<td>130000000000</td>
</tr>
<tr>
<td>HAND</td>
<td>RIGHT</td>
</tr>
<tr>
<td>HELWNGLE</td>
<td>30.0000000000</td>
</tr>
<tr>
<td>FILLETRAD</td>
<td>0.0000000000</td>
</tr>
<tr>
<td>INNERIA</td>
<td>716.0000000000</td>
</tr>
<tr>
<td>ROOTDIA</td>
<td>205.0000000000</td>
</tr>
<tr>
<td>GLUTERIA</td>
<td>214.0000000000</td>
</tr>
<tr>
<td>BACKSHOULDERIA</td>
<td>205.0000000000</td>
</tr>
<tr>
<td>VOLUNSMOD</td>
<td>20610900000000</td>
</tr>
<tr>
<td>RADIUSDEFINING</td>
<td>2</td>
</tr>
<tr>
<td>POSISSION</td>
<td>0.2900000000</td>
</tr>
<tr>
<td>DENSITY</td>
<td>7.0100000000+009</td>
</tr>
<tr>
<td>ALPHA</td>
<td>0.0010000000</td>
</tr>
<tr>
<td>BETA</td>
<td>1.0000000000+007</td>
</tr>
</tbody>
</table>

Figure 6.13: The ring gear tooth menu.
Table 6.1: FE Rim RACE Menu Input Data

<table>
<thead>
<tr>
<th>Race</th>
<th>DIA (mm)</th>
<th>AXPOSN1 (mm)</th>
<th>AXPOSN2 (mm)</th>
<th>CIRCORDER</th>
<th>AXIALORDER</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>214.00</td>
<td>0.00</td>
<td>15.00</td>
<td>32</td>
<td>2</td>
</tr>
<tr>
<td>2</td>
<td>229.00</td>
<td>13.00</td>
<td>15.00</td>
<td>32</td>
<td>1</td>
</tr>
</tbody>
</table>

6.3 Ring Rotor Shafts

The ring gear rims of the ring1 and ring2 rotors are imported into Transmission3D as an FE mesh created by an external FE software package. We demonstrate the process for doing so in this section. This method is recommended for complex rim components or if a 3D CAD model of the rim is readily available. The process presented is for the case of the rim of the ring2 rotor however, the general process for the ring1 rotor rim is the same. The differences that exist are due to the symmetry of the rim which results in a different orientation and position of the components.

Figure 6.14 shows the new rim model which includes the FE rim mesh and two shafts. Shaft 1 is identical to shaft 2 from the previous method. We keep this shaft separate due to the contact condition applied on the back side. The contact surfaces feature for the FE carrier is limited to planar and cylindrical surfaces. In this case, the surface is conical with a 90 degree cone angle so we simply attach a shaft and use the FRONTCONTACT and BACKCONTACT features within the SHAFT menu as explained in the previous section. Shaft two is included for the purpose of applying the shaft constraint. Alternatively, we could have chosen to apply this constraint to the nodes on the surface of the FE mesh within the external FE software.

Figure 6.15 shows the external mesh of the rim model. We take advantage of the axial symmetry of the rim by creating an FE mesh of a small sector. We import only the sector and tell Transmission3D how many sectors are required to complete the full rim model. The model can be created using any commercially available package which exports files in Nastran (.bdf) format. We use the same cross-sectional dimensions given for the rim FE mesh drawing in Figure 6.14 to create the FE model. Using these dimensions, we ensure the origin of the rim matches the rotor origin. Doing so simplifies the importing process. We export the 3D mesh elements of the rim sector model in Nastran format.

6.3.0.1 The Carrier Menu

Selecting the ENABLECARRIERS checkbox in the rotor 3 menu enables the carrier menu (Figure 6.18). We use the carrier menu to import the FE mesh of the rim. We select TYPE = FECARRIER_NASTRAN in order to import the finite element mesh file (.bdf) created in the previous section. For this example, we use only one file so NFECARRIERFILES=1. NODETOLERANCE sets the maximum tolerance used to fine nodes at race locations. We set the tolerance to 0.001 in this case. MAXJOINTANGLE specifies the maximum angle that should be used to smooth joining shell elements. Anything greater that the specified angle is left as a sharp corner. We use the recommended default value of 15 degrees.

In modeling the FE rim segment, we oriented the rim such that its origin aligns with rotor 3 origin. We set AXILSHIFT to 0 since the origins are coincident. ENABLEPINIONS is set to OFF since we are not modeling a traditional carrier. This allows us the option to import FE shafts and rims using the FECARRIER TYPE. As mentioned earlier, we modeled only a small sector of the rim since the remaining portion can be modeled with identical copies revolved about the rotational axis. We select USESECTORALSYMMETRY to take advantage of this feature. Doing so greatly reduces computer time during preprocessing and when running an analysis. NGROUPS represents the number of sectors when ENABLEPINIONS=FALSE. We set NGROUPS = 72 since we are importing a 5 degree sector, resulting in the 72 * 5 = 360 degree rim desired.

We set NRACES=2 in order to connect the two shafts. The locations of the two races are specified within the RACE menu. The RACE menu input parameters are provided in Table 6.1. The FILE menu is used to enter the information for each carrier file. For the one file used in this example, we enter: FEFILENAME=RingRim2.bdf, SUBTREENAME=Rim2, and PREFERREDCUTTINGDIR=Z.

An identical process can be used to create the RingRim1 rotor. Refer to Figure 6.15 to set up the shaft segments and use the Nastran file RingRim1.bdf located within the SAMPLES/DoubleHelicalTwoRingsJournals subdirectory.
Figure 6.14: The ring2 rotor rim shafts using rim FE mesh.

Figure 6.15: The ring2 rotor rim FE mesh model.
Figure 6.16: The shaft menu for shaft 2.

Figure 6.17: The segment menu for segment 1 of shaft 2.
Figure 6.18: The rotor 3 carrier menu using the FECARRIER_NASTRAN carrier option.
6.3.0.2 The Shaft Menu

Within the shaft menu, we now model two shafts to complete the rim. Shaft 1 of the FE mesh rim model is identical to shaft 2 using the previous method of compound shafts. The shaft details and segment menu are given in Figures 6.14 and 6.19, respectively.

The second shaft using the rim FE mesh method is a small single-segment shaft as shown in Figure 6.14. The AXIALPOSN is entered as 13.00 within the shaft menu. The segment menu for shaft 2 is provided in Figure 6.20. We apply the rigid shaft constraint on the outer diameter of the shaft by checking ODCONSTRAINED and selecting TYPE = RIGID. The inside diameter of shaft 2 contains a race surface for joining with the FE rim. CIRCORDERINNER = 32 and AXIALORDERINNER = 1 are the same values used in the RACE menu of the corresponding FE rim race.

We proceed with this example using the FE mesh method for the ring rims of rotors 2 and 3. All subsequent menus and results presented in this example assume this method is used.
Figure 6.20: The shaft 2 segment menu menu.
6.4 The Carrier Rotor

The Carrier rotor, shown in Figure 6.21, is the fourth rotor of the Double-Helical Planetary model. The carrier rotor consists of a shaft (ENABLESHAFTS = TRUE, NSHAFTS = 1) and a carrier (ENABLECARRIERS = TRUE, NCARRIERS = 1). The carrier in is made up of the following components: a common race, shoulder, pinion pin, carrier race, and four pinion groups with two decks. The carrier rotor origin is located at XPOS = 0, YPOS = 0, ZPOS = 0 with its rotational axis direction set as AX = 0, AY = 0, AZ = 1. The rotor TYPE is INPUT with RPM = -1000. The carrier rotor is supported by stiffness bearings so we do not constrain the rotor reference frame. No loads are applied to this rotor.

6.4.1 The Carrier Rotor Shaft

The carrier rotor shaft is modeled in the shaft submenu within the rotor menu. Figure 6.22 shows the shaft details. In the shaft menu we set AXIALPOSNSHAFT = 20. The shaft is modeled using 4 segments (NSEGMENTS = 4). The shaft segment inputs are entered into the segement menu shown in Figure 6.23. The required inputs for the remaining shaft segments are given in Figure 6.22. The outside diameter of the first shaft segment joins to the inside diameter of the carrier race, so we check ODRACE. The inside diameter surfaces of segments 1 and 2 connect to the stiffness bearing races, so we check IDRACE for these segments. We set ODCONSTRAINT = FLEXIBLE on segment 4 to keep the stiffness matrix non-singular.

6.4.2 The Carrier

The carrier is modeled in the carrier menu. The carrier includes the carrier race, common shaft, shoulder, pinion pin, and the pinion groups. The carrier TYPE = RIGID, which uses an infinite stiffness for the connection between the pinion races and the shaft race. We set NRACES = 1 to connect the carrier to the carrier shaft. Checking ENABLEPINIONS allows us to model the pinions by enabling the group and pinion submenus. The carrier contains four copies of one pinion, so we set NPINIONS = 1 and NGROUPS = 4.
Figure 6.22: The Carrier rotor shaft details.

Figure 6.23: The Carrier rotor shaft segment menu.
Table 6.2: The group menu inputs.

<table>
<thead>
<tr>
<th>Group</th>
<th>AXIALPOSN</th>
<th>THETA</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0.00</td>
<td>0</td>
</tr>
<tr>
<td>2</td>
<td>0.00</td>
<td>90</td>
</tr>
<tr>
<td>3</td>
<td>0.00</td>
<td>180</td>
</tr>
<tr>
<td>4</td>
<td>0.00</td>
<td>270</td>
</tr>
</tbody>
</table>

The Carrier Race  The race menu allows us to define a race surface in order to connect an output shaft. The race in this model connects to segment 1 of the carrier rotor shaft, as shown in Figure 6.22. AXPOSN1RACE and AXPOS2RACE are the distances from the rotor origin of the front and back sides of the race, respectively. We enter AXPOSN1RACE = 20 and AXPOS2RACE = 30. DIARACE = 40 sets the race diameter to be the same as the diameter of the mating surface on the carrier shaft segment 1. CIRCORDER = AXIALORDER = 2 matches the interpolation order of the mating surface of the shaft.

The Carrier Pinion Groups  The pinion group positions are defined within the group menu. Table 6.2 shows the inputs used for this example. We have four groups, or copies, of one pinion each separated by an angle of 90 degrees. Each group origin is located on the same x-y plane coincident with the rotor origin by setting AXIALPOSN = 0.

The Carrier Pinion  The carrier pinion inputs are entered into the pinion submenu (6.25). Within the pinion menu, the PINPOSNERROR, PIN, and DECK menu options are available. The PINPOSNERROR menu allows the users to model pin misalignment. We do not use this feature in this example and all error values are left to the default value of 0. The pinion menu is used to enter the general pinion information such as the number of decks, pin type, and pinion position. If the number of decks is greater than 1, the common race and shoulder input options appear in the pinion menu. Figure 6.24 shows the details of pinion deck 1.

The Pinion Pin  The planetary pinion pin is modeled in the pin submenu of the pinion menu (Figure 6.24). The pin TYPE = SIMPLE is set within the pinion menu. A simple pin is one that contains just a single segment. The COMPOUND option can be used for pins that require multiple segments.

The Pinion Deck  The pinion deck menu is used to enter the inputs specific to each individual pinion deck. The RIM, TOOTH, SPACEERR, and LEADERR submenus are available within the pinion menu, but we use only the TOOTH menu in this example. The DECK input changes the pinion deck number for which the inputs are entered. The tooth menu for deck 1 is displayed in Figure 6.27. We set AXIALPOSN = -8.5 to shift the deck origin in the direction of the rotor rotational axis with respect to the pinion group origin. The tooth design parameters are also set in this menu.

The second pinion deck is modeled by setting DECK = 2 in the pinion menu. The tooth submenu for the second deck is enabled upon doing so. Deck 2 is identical to deck 1 with the exception of two differences. The first is the second pinion deck origin is shifted in the opposing direction of the first by entering AXIALPOSN = 8.5. The second is the difference in hand, with the second deck being right handed (HAND = RIGHT). The remaining inputs of Figure 6.27 remain the same.
Figure 6.24: The carrier pinion details.
Figure 6.25: The carrier pinion menu.
A DOUBLE-HELICAL PLANETARY SYSTEM

Figure 6.26: The carrier pinion pin menu.

Figure 6.27: The carrier pinion tooth menu.
6.5 Connectors

The Double-Helical Planetary model includes 5 connectors, which are modeled in the connectors menu within the edit menu. Connectors 1 and 2 are stiffness bearings that connect the sun rotor shaft to ground. Connectors 3 and 4 are stiffness bearings that connect the carrier shaft to ground. The fifth connector is the pinion journal bearing.

None of the connectors in this example are physical bearing models. The stiffness bearings are modeled using the stiffness matrix diagonals to approximate a bearing connection between the rotor and a rigid ground. No detailed contact information is available for the stiffness connector type. The pinion bearing is a journal type connector. A journal connector merely defines a contact surface in order to constrain radial motion and obtain pressure data at the interface.

6.5.1 Stiffness Bearings

The connector menu for the first sun rotor stiffness bearing is shown in Figure 6.28. This connector joins segment 1 of the sun rotor shaft to ground so we set MEMBER1TYPE = ROTOR, ROTOR =1, and MEMBER2TYPE = GROUND. The bearing origin position and positive rotational axis are set by XPOS = YPOS = 0, ZPOS = 5.00 and AX = AY = 0, AZ = 1, respectively. The outer race is designated as race 1 by setting DIARACE1 = 40.00. The distances of the back and front of the inner bearing race relative to the bearing origin are AXPOSN1RACE1 = -10.00 and AXPOSN2RACE1 = 10.00, respectively. The inner race is designated race 2 by setting DIARACE2 = 30.00. The back and front of the inner race are set in the same way as the outer, with AXPOSN1RACE2 = -5.00 and AXPOSN2RACE2 = 5.00. STANDARD = TRUE enables the standard stiffness bearing menu items KR, KZ, KTHETAR, KTHETAZ, CR, CZ, CTHETAR, and CTHETAZ. The stiffness values used are shown in Figure 6.28.

The connector menus for stiffness bearing type connectors 2, 3, and 4 require the same inputs as connector 1. Table 6.3 shows the bearing position parameters used for each. Each stiffness connector’s rotor rotational axis in the positive z-direction (AX = AY = 0, AZ = 1) and uses the same stiffness values as those of connector 1.
Table 6.3: The stiffness bearing connector race positions.

<table>
<thead>
<tr>
<th>Connector</th>
<th>XPOS</th>
<th>YPOS</th>
<th>ZPOS</th>
<th>AXPOS1 RACE2</th>
<th>AXPOS2 AP2R1</th>
<th>DIA RACE1</th>
<th>AXPOS1 RACE2</th>
<th>AXPOS2 RACE2</th>
<th>DIA RACE2</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0.00</td>
<td>0.00</td>
<td>5.00</td>
<td>-10.00</td>
<td>10.00</td>
<td>40.00</td>
<td>-5.00</td>
<td>5.00</td>
<td>30.00</td>
</tr>
<tr>
<td>2</td>
<td>0.00</td>
<td>0.00</td>
<td>75.00</td>
<td>-15.00</td>
<td>15.00</td>
<td>40.00</td>
<td>-5.00</td>
<td>5.00</td>
<td>30.00</td>
</tr>
<tr>
<td>3</td>
<td>0.00</td>
<td>0.00</td>
<td>100.00</td>
<td>-5.00</td>
<td>5.00</td>
<td>30.00</td>
<td>-5.00</td>
<td>5.00</td>
<td>20.00</td>
</tr>
<tr>
<td>4</td>
<td>0.00</td>
<td>270</td>
<td>140.00</td>
<td>-5.00</td>
<td>5.00</td>
<td>30.00</td>
<td>-5.00</td>
<td>5.00</td>
<td>20.00</td>
</tr>
</tbody>
</table>

Figure 6.29: The connector menu for the pinion journal bearing.

6.5.2 Pinion Journal Bearing

The connectors menu for connector 5 is shown in Figure 6.29. The journal bearing defines a contact surface between the pinion pin and the common race components, constraining the radial motion of the pinion. Within the connectors menu we set CONNECTOR = 5 and TYPE = JOURNAL. Transmission3D automatically creates the contact surface at the common race-pinion pin interface when MEMBER1TYPE = PINION is selected. The pinion is identified by setting IDROTOR1 = 4, IDCARRIER = 1, and IDPINION = 1. The bearing origin is coincident with the pinion rotor origin (AXIALPOS = 0) and the radial clearance is set to 0.4.
Six contact pairs are used in this example. The first four are gear contacts which define the sun1-deck1, sun2-deck2, ring1-deck1, and ring2-deck2 gear pairs. The fifth contact pair defines the shaft-shaft contact between the two shafts on which we defined front and back contact surfaces.

The sun1-deck1 pair menu inputs identify the two contacting components and define the size of the contact grid. To define the sun1-deck1 pair, we select TYPE = SUN_PINION. Next, we identify the sun-pinion pair by setting the sun to sun1 (IROTOR1 = 1, ISUN1 = 1). The pinion is identified as pinion deck 1 by selecting IROTOR2 = 4, ICARRIER2 = 1, IPINION2 = 1, and IDECK2 = 1.

The contact grid size is specified by setting the number and spacing of the grid point divisions in the profile and facewidth directions. Here, we set the number of divisions between grid points on each side of the center grid point to be 2 in the profile direction (NPROFDIVS = 2) and 7 in the facewidth direction (NFACEDIVS = 7). The grid point spacing is set automatically in the facewidth direction, since the grid spans the entire width of the face (from T = -1 to T = 1) and the number of division is entered. We choose the spacing in the profile direction as DSPROF = 0.04. SEPTOL = 0.1 sets the maximum distance of separation for contact surfaces that is considered for contact. Figure 6.31 shows the pair menu inputs for the sun1-deck1 contact pair.

The shaft-shaft contact pair is defined simply by identifying the rotor, shaft, segment, and segment surface for each contacting surface, as well as the separation tolerance. The inputs for the shaft-shaft contact pair are given in Figure 6.31.
Figure 6.31: The pair menu for the shaft-gear contact.
6.7 Analysis Setup

The setup and range menus are given in Figures 6.32 and 6.33. We begin the analysis at INITIALTIME = 0 and write the results to the POSTFILENAME = postproc.dat by checking POSTPROCWRITE. We set the SOLNMETHOD = STATIC within the range menu and run the analysis for NTIMESTEPS = 11. The following equations are used to calculate the time for one tooth mesh cycle ($\Delta T_{\text{cycle}}$) and DELTATIME between each time step ($\Delta t$).

$$\Delta T_{\text{cycle}} = \frac{2\pi}{|\omega_{\text{ring}} - \omega_{\text{carrier}}| N_{\text{ring}}}$$

$$\Delta t = \frac{T_{\text{cycle}}}{NTIMESTEPS-1}$$

where: $\omega_{\text{ring}} = 0$ rad/s

$\omega_{\text{carrier}} = -104.72$ rad/s

$N_{\text{ring}} = 100$
Figure 6.33: The analysis range menu.
6.8 Results

6.8.1 Rim Contact Pressure

The contact pressure distribution on the ring 1 rim is shown in Figure 6.34.

6.8.2 Ring Gear Deflection

The deflection of the ring 1 gear is determined using the point displacement postprocessing menu (Figure 6.35). We choose a point on the base surface of the gear tooth mesh as shown in Figure 6.36 and execute the postprocessing menu for each tooth of the ring 1 gear. So we choose MESH = TOOTH, TOOTHBEGIN = 1, and TOOTHEND = 100 in the point displacement menu. The element number at the point of interest is displayed in the vertex info box within the IGlass file (Fig. 6.36). We may choose any of the four elements listed since they all share the node of interest. In this case, we choose ELEM = 112. The COORD_TYPE = ELEM_COORDS with XI = 1, ETA = -1, and ZETA = 1 as shown in the vertex info box for element 112. The displacement data is output to the a file by checking OUTPUTTOFILE and specifying an output FILENAME.

A MATLAB script file named PlotRingGearDeflection.m is located in the subdirectory SAMPLES/DoubleHelicalTwoRingsJournals. This program can be used to plot the ring gear deflection in the radial, tangential, and axial directions. The point displacement output file Ring1PointDisplacement.txt is also located in the subdirectory and is required to run the MATLAB file. The output file must be altered slightly in order for the MATLAB program to execute properly.
The time data (column 1) and position and deflection data about the body origin (columns 2 through 7) of the original output file have been removed. The program uses the position data to shift the average position of the gear base surface to (0,0,0). The reference frame is then transformed into radial and tangential components by multiplying the UX and UY data by the transformation matrix included in the program. The resulting plots are provided in Figures 6.37 through 6.39.
Figure 6.37: The radial deformation on the ring I rotor gear.
Figure 6.38: The tangential deformation on the ring 1 rotor gear.
Figure 6.39: The axial deformation on the ring 1 rotor gear.
The point strain postprocessing menu shown in Figure 6.40 returns strain data at a location and direction specified by the user. Here we will demonstrate how to use the point strain postprocessing menu to gather strain data in the tangential direction around the circumference of the ring 1 rotor rim.

The first two inputs of the point strain menu define the BODY on which we would like to gather strain data and the coordinate system (COORD_TYPE) we would like to use to identify the nodes to sample. We select BODY = RING1_ROTOR and COORD_TYPE = ELEM_COORDS in this case. ELEM_COORDS is the element coordinate system, which is the reference frame attached to each individual finite element. Selection of this coordinate system formats the point strain menu as shown in Figure 6.40. We set TOOTHBEGIN = TOOTHEND = 1 to define sector 1 as the location of the node of interest. The first rim sector is shown in Figure 6.41. The remainder of the gear rim is created from identical copies of sector 1 so we first explain the point strain menu inputs for this sector.

The MESH input of the point strain menu describes the submesh on which the node of interest is located. The submeshes are created by Calyx when we import the rim as an external FE mesh in order to reduce the computer time required to convert the complete mesh. Calyx then 'rebuids’ the complete mesh out of the submeshes when an analysis of the model is performed. We then select the location of the nodes where we wish to gather the strain information. Figure 6.42 shows the nodes we have chosen on sector 1, along with the element information. The nodes are centrally located on the outside surface of the elements. We discuss the point strain menu inputs for the node on sector 1 at element 10 here. Later, we discuss the process for gathering the strain data for the remaining elements and sectors of the rim.

The information shown in Figure 6.42 can be obtained in the “vertex info” box under the “attributes” tab of a preprocessing IGlass file of the model by double-clicking on the node of interest. We set MESH = RINGRIM1_SECTOR_2_1_1_2_1_1_2_1, ELEM = 10, XI = 0, ETA = -1, and ZETA = 0 to define the location of the node on element 10 shown in Figure 6.42.

We set the direction in which we would like to measure the strain with the DIR_COEFF_XI, DIR_COEFF_ETA, and DIR_COEFF_ZETA inputs. These are the direction cosines of the angles between the direction in which we would like to measure the strain and the Xi, Eta, and Zeta axis, respectively. In order to determine these values, we first must determine the orientation of the element reference frame at the element of interest. Figure 6.43 shows element 10, along with the “vertex info” for each corner node of the outer surface. The element coordinate system origin is located at the geometric center of the origin. Each face surface of the element is located at a distance of 1 from this origin. Using this information and the coordinates of each corner point on the outer surface of the element we can determine the orientation of the element reference frame to be as is shown. Since we would like to output strain data in the tangential direction, we set DIR_COEFF_XI = 0, DIR_COEFF_ETA = 0, and DIR_COEFF_ZETA = 1.

Up to this point we have discussed the process for obtaining strain data for a single point. However we would like to obtain this data for the entire circumference of the ring gear rim for one complete mesh cycle (11 time steps). We do so by creating a postprocessing script which runs the post processing menu for the five elements of sector 1, followed by the same five elements of each subsequent sector for the first time step. We then repeat this process for the remaining 10 time steps. The MATLAB program Ring1RimPointStrainScriptSetup.m writes the Multyx script file RINGRIM1_POINTSTRAIN_SCRIPT.TXT. Executing the Multyx script results in the output file Ring1RimPointStrain.txt containing the tangential strain data for the nodes of each sector of the rim for one complete mesh cycle. The MATLAB file PlotPointStrain.m reads the output file and returns the plot shown in Figure 6.44. All of the abovementioned files are provided in the subdirectory SAMPLES/DoubleHelicalTwoRingsJournals within the default working directory.
Figure 6.40: The point strain postprocessing menu.
Figure 6.41: Ring 1 rim sector 1.
Figure 6.42: The nodes at which the strain data is collected.
Figure 6.43: The element coordinate system.
Figure 6.44: The tangential strain on the ring1 rotor rim.
In this chapter we demonstrate how to model the Helical Reduction System model with a housing. We describe how to model the housing using both the FE mesh and condensed stiffness matrix methods and explain the advantages of each method. We also show how to include misalignment resulting from shaft or housing manufacturing errors. The session file titled RollingElemBrgWithHousing.ses can be found in the subdirectory SAMPLES/ReductionSetWithHousing within the default directory. We use Newtons as the unit of force, millimeters as the unit of length, and seconds as the unit of time in this example.

The reduction system consists of three rotors: the Input Rotor, Idler Rotor, and Output Rotor. The input and output rotors are each modeled with a shaft and a helical gear. The idler rotor is made up of a shaft and two helical gears. We define torque at the output rotor and speed at the input rotor. We set the speed of the input rotor by choosing TYPE = INPUT and RPM = -3.33 within the edit menu for rotor 1. The torque is defined at the output rotor by setting TYPE = OUTPUT, TORQUE = 1000000.
Figure 7.1: The Helical Reduction FE Model.
7.1 The Input Rotor

We model the input rotor within the rotor menu by setting ROTOR = 1. The rotor menu for this rotor is provided in Figure 7.3. The rotor is made up of an input shaft and a helical gear as shown in Figure 7.2. Setting ENABLESHAFTS = TRUE, NSHAFTS = 1 and ENABLESUNS = TRUE, NSUNS = 1 enables the shaft and sun submenus. The rotor origin is coincident with the origin of the global reference frame so we set XPOS = YPOS = ZPOS = 0. We choose the positive z-axis as the rotational axis of the input rotor by setting AX = AY = 0 and AZ = 1. We define the speed at the input rotor by choosing TYPE = INPUT. The speed of the rotor is RPM = -3.3333. The direction of rotation follows the left hand rule about the rotational axis since the sign of the speed is negative. The rotor reference frame will be sufficiently constrained by two bearings so there is no need to constrain the rotor reference frame. Thus, we set UXCONSTRAINT = UYCONSTRAINT = UZCONSTRAINT = THETAXCONSTRAINT = THETAYCONSTRAINT = FALSE. We do not wish to apply a concentrated load at the rotor origin so we leave FX, FY, FZ, MX, and MY to the default value of 0.

7.1.1 Input Shaft

The input rotor shaft details are provided in Figure 7.4. The menu inputs for the shaft are entered into the shaft menu (Figure 7.5). We shift the starting point of the first shaft segment along the rotational axis by entering AXIALPOSNSHAFT = -90. The shaft is modeled using 8 segments so we set NSEGMENTS = 8. The material properties shown for the input shaft are used consistently throughout this example for each component and will not be provided from this point on. Checking UNIFORMMATERIAL applies these same material properties to each shaft segment automatically.

The individual shaft segments are modeled in the segment menu. The segment menu inputs are provided within Figure 7.4. The menu for segment 1 is provided in Figure 7.6 as a reference. The input shaft consists of three races, defined by setting ODRACE = TRUE within the segment 3, 5, and 7 menus. The order of interpolation between elements in the circular and axial directions are set to CIRCORDER = 8 and AXIALORDER = 2, respectively, for each race surface. Shaft segment 1 contains the shaft constraint condition required to keep the stiffness matrix non-singular. We do so by checking ODCONSTRAINED within the segment 1 menu. We set the constraint type to FLEXIBLE and use the same orders of interpolation as for the race conditions.

7.1.2 Input Sun Gear

The input rotor sun gear, shown in Figure 7.7, is modeled within the sun menu. The sun menu inputs are provided in Figure 7.8. The sun origin coincides with the rotor origin, so we set AXIALPOSN = 0 within the sun menu. The sun submenus are also shown in Figure 7.8. Within the base menu, we define the race at the base surface of the gear. The order of interpolation between elements must match those at the mating shaft surface so we set CIRCORDER = 8 and AXIALORDER = 2. The tooth submenu is the only other menu used for the input rotor sun gear. We set HELIX angle
Figure 7.3: The input rotor menu.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>NAME</td>
<td>INPUT</td>
</tr>
<tr>
<td>XPOSN</td>
<td>0.0000000000e+000</td>
</tr>
<tr>
<td>YPOSN</td>
<td>0.0000000000e+000</td>
</tr>
<tr>
<td>ZPOSN</td>
<td>0.0000000000e+000</td>
</tr>
<tr>
<td>AX</td>
<td>0.0000000000e+000</td>
</tr>
<tr>
<td>MX</td>
<td>0.0000000000e+000</td>
</tr>
<tr>
<td>RPM</td>
<td>-3.3333333333</td>
</tr>
<tr>
<td>NSHAFTS</td>
<td>1</td>
</tr>
</tbody>
</table>

Figure 7.4: The input shaft details.

ODRACE = 8
CIRCORDER = 8
AXIALORDER = 2

ODRACE = 8
CIRCORDER = 8
AXIALORDER = 2

ODRACE = 8
CIRCORDER = 8
AXIALORDER = 2

ODCONSTRAINT = FLEXIBLE
CIRCORDER = 8
AXIALORDER = 2

Z

SEG8 5.00
SEG7 15.00
SEG6 55.00
SEG5 40.00
SEG3 15.00
SEG2 20.00
SEG1 25.00
SEG4 10.00

Rotor Axis = (0, 0, 1)
AXIALPOSNSHAFT = -90.00
Figure 7.5: The input shaft menu.
Figure 7.6: The input shaft segment 1 menu.
Figure 7.7: The sun gear details.

= 18 to and HAND = LEFT to define the gear type as helical. The remaining tooth menu inputs are provided in Figure 7.9.
Figure 7.8: The sun gear menu.
Figure 7.9: The sun gear tooth menu.
7.2 The Idler Rotor

The idler rotor (Figure 7.10) is the second rotor of the reduction model. We model this rotor within the rotor menu by setting ROTOR = 2. The rotor origin is located at 72 mm above the global origin along the y-axis, so we set XPOS = ZPOS = 0 and YPOS = 72. The idler rotor rotational axis is also oriented in the positive z-direction so we set AX = AY = 0, and AZ = 1. Setting ENABLESHAFTS = ENABLESUNS = TRUE, NSHAFTS = 1, and NSUNS = 2 enables the shaft and sun menus allowing us to model the shaft and two sun gears contained on the rotor. As the rotor name implies, we set TYPE = IDLER. The idler rotor is sufficiently constrained by two bearings, so we leave the rotor constraint boxes unchecked. No loads are applied to the rotor.

7.2.1 Idler Shaft

The idler rotor shaft details are given in Figure 7.11. The shaft is shifted along the rotor rotational axis by AXIALPOSNSHAFT = -55.00. The shaft consists of nine segments (NSEMENTS = 9), which are modeled within the segment menu. The segment details are covered sufficiently within Figure 7.11. Figure 7.12 shows the segment 1 menu inputs as a reference. The shaft constraint is applied to the outside diameter of segment 2 and is of ODTYPE = FLEXIBLE. The orders of interpolation for the flexible constraint and all shaft races are CIRORDER = 8 and AXIALORDER = 2.

7.2.2 Idler Sun Gears

The idler rotor contains two sun gears. The first gear is coincident with the rotor origin as shown in Figure 7.13. Within the base submenu, we set CIRCORDER = 8 and AXIALORDER = 2 to match the order of interpolation on shaft segment 4 where the gear connects. The tooth menu containing the gear tooth design parameters for sun 1 is provided in Figure 7.14.

The second idler gear is located at an axial distance of AXIALPOSN = 45.00 from the rotor 2 origin. Figures 7.15 and 7.16 provide the necessary details for setting up the base and tooth menus for sun 2.
Figure 7.11: The idler shaft details.

Figure 7.12: The idler shaft segment 4 menu.
Figure 7.13: The idler sun gear 1 details.
Figure 7.14: The sun gear 1 tooth menu.

<table>
<thead>
<tr>
<th>MODRN</th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>EXIT</td>
<td>QUIT</td>
</tr>
<tr>
<td>NTBTH</td>
<td>84</td>
<td></td>
</tr>
<tr>
<td>NPACELEMS</td>
<td>41</td>
<td></td>
</tr>
<tr>
<td>COORDORDER</td>
<td>10</td>
<td></td>
</tr>
<tr>
<td>DISPLORDER</td>
<td>3</td>
<td></td>
</tr>
<tr>
<td>PROTOTYPE</td>
<td>SIMPLE</td>
<td></td>
</tr>
<tr>
<td>SYMMETRICTOOTH</td>
<td></td>
<td></td>
</tr>
<tr>
<td>PLANE</td>
<td>TRANSVERSE</td>
<td></td>
</tr>
<tr>
<td>XVERSEDAMPITCH</td>
<td>0.5000000000</td>
<td></td>
</tr>
<tr>
<td>XVERSEPRESANGLE</td>
<td>0.0300000000</td>
<td></td>
</tr>
<tr>
<td>XVERSETHICKNESS</td>
<td>0.0100000000</td>
<td></td>
</tr>
<tr>
<td>FACEWIDTH</td>
<td>0.0000000000</td>
<td></td>
</tr>
<tr>
<td>HAND</td>
<td></td>
<td>RIGHT</td>
</tr>
<tr>
<td>HELCRANGL</td>
<td>15.0000000000</td>
<td></td>
</tr>
<tr>
<td>PACTPRAD</td>
<td>0.0000000000</td>
<td></td>
</tr>
<tr>
<td>ROOTDIA</td>
<td></td>
<td>39.0000000000</td>
</tr>
<tr>
<td>INNERRDIA</td>
<td></td>
<td>32.0000000000</td>
</tr>
<tr>
<td>OUTERDIA</td>
<td></td>
<td>48.0000000000</td>
</tr>
<tr>
<td>RIMDIA</td>
<td></td>
<td>96.0000000000</td>
</tr>
<tr>
<td>DOCHAMFER</td>
<td></td>
<td></td>
</tr>
<tr>
<td>ENABLEFRONTDOFSET</td>
<td></td>
<td></td>
</tr>
<tr>
<td>ENABLEBACKDOFSET</td>
<td></td>
<td></td>
</tr>
<tr>
<td>YOUNGSMOD</td>
<td></td>
<td></td>
</tr>
<tr>
<td>POISON</td>
<td></td>
<td></td>
</tr>
<tr>
<td>DENSITY</td>
<td></td>
<td></td>
</tr>
<tr>
<td>ALPHA</td>
<td></td>
<td></td>
</tr>
<tr>
<td>BETA</td>
<td></td>
<td></td>
</tr>
<tr>
<td>TEMPLATE</td>
<td></td>
<td>MEDIUM TPL</td>
</tr>
</tbody>
</table>

Figure 7.15: The idler sun gear 2 details.
Figure 7.16: The sun gear 2 tooth menu.
7.3 The Output Rotor

The final rotor of the helical gear reduction model is the output rotor. This rotor is comprised of a shaft (ENABLE-SHAFTS = TRUE, NSHAFTS = 1) and a helical gear (ENABLESUNS = TRUE, NSUNS = 1). The rotor origin is located at XPOS = ZPOS = 0, YPOS = 166.00. The positive rotational axis of the output shaft is the z-axis (AX = AY = 0, AZ = 1). We would like to define the torque at this rotor so we set TYPE = OUTPUT and TORQUE = 1000000. Two bearings constrain the rotor so we leave all constraint boxes unchecked. No loads are applied to the rotor.

7.3.1 Output Shaft

The output shaft details are provided in Figure 7.18. The shaft is shifted along the rotational axis by AXIAL-POSNSHAFT = -58.00. Eight segments are used to build the shaft and are modeled within the segment menu. The segment menu for segment 8 is provided in Figure 7.19. The shaft constraint is applied to the outer diametral surface of segment 8 by checking ODCONSTRAINED. We set ODTYPE = FLEXIBLE, CIRCORDER = 2, and AXIALORDER = 2 for the constraint condition. ODRACE is checked for segments 2, 4, and 6 in order to connect the gear and two connectors. We set CIRCORDER = 8 and AXIALORDER = 2 for each of these race conditions.

7.3.2 Output Sun Gear

The output sun gear details are given in Figure 7.20. The gear mid-face is shifted by an AXIALPOSN = 45.00 from the rotor origin. The base surface of the gear mates with segment 4 of the output shaft so we set CIRCORDER = 8 and AXIALORDER = 2 within the base menu. The tooth design parameters are entered into the tooth menu given in Figure 7.21.
Figure 7.18: The output shaft details.

Figure 7.19: The output shaft segment 8 menu.
Figure 7.20: The output sun gear details.
Figure 7.21: The sun gear tooth menu.
7.4 The Housing

The housing model is described here using two different methods. With the first method, we import the housing as an FE mesh from a model created using external FE software. The second method described is the condensed stiffness method. With this method we condense the housing FE model into a stiffness matrix of 10 nodes. Six equivalent stiffness components are then calculated at each node in order to approximate the overall housing stiffness. This method is much more time efficient when analyzing a model and is recommended when the housing is simple in geometry and relatively rigid.

7.4.1 The FE Housing

The FE housing model is pictured in Figure 7.22. We create the model using an external FE software package and export the mesh in Nastran (.bdf) file format. The FE model reference frame origin is coincident with the rotor origin, and the orientation of each is also identical.

Figure 7.24 shows the housing menu for the FE housing. The housing TYPE = FEHOUSING_NASTRAN, and we fully constrain the housing reference frame by checking UXCONSTRAINT, UYCONSTRAINT, UZCONSTRAINT, THETAXCONSTRAINT, THETAYCONSTRAINT, and THETAZCONSTRAINT. XSHIFT, YSHIFT, and ZSHIFT are used only if the housing and global reference frames differ, so we set all three to zero in this case.

The housing RACE menu is shown in Figure 7.25. The origin of each race is entered using the XPOS, YPOS, and ZPOS inputs, and the positive rotational axis vector direction is defined with the AX, AY, and AZ inputs. AXPOSN1 and AXPOSN2 set the axial distances of the race ends from the origin. The model contains 6 housing races, one at each hole location on the housing. Figure 7.23 can be used to obtain the race menu inputs for each of the 6 races.
Figure 7.23: The FE housing details.

Figure 7.24: The FE housing menu.
Figure 7.25: The FE housing race menu.
7.4.2 Condensed Housing Stiffness Matrix

We now describe the condensed stiffness method for modeling the housing component of the reduction system. With the condensed stiffness method, we condense the FE housing into a stiffness matrix at 10 nodes, with 6 degrees of freedom at each node (60 x 60 matrix). We connect the outside diametral surfaces of the shaft bearing races to 6 of these nodes, and constrain the remaining 4 to model a bolted connection at the housing base. The details of the FE model used for this method are given in Figure 7.26. Note the housing origin is now located at one of the bearing mid-points on the output shaft and the housing reference frame orientation differs from that of the housing used for the FE housing method (Figure 7.22).

7.4.2.1 HyperMesh Condensed Stiffness Matrix Tutorial

In order to generate the bulk data file of the stiffness matrix which is able to be imported into Transmission3D, we must first prepare the model to be analyzed by the mesh condensing program. This is accomplished using most any FE software package. Here, we describe the process for doing so using HyperMesh. Radioss, a sub-program of HyperMesh, is then used to condense the model into the 60 x 60 matrix and output the bulk data file.

1. The FE model is imported into HyperMesh using the Radioss Bulk Data profile. The user profile window is displayed on the screen when HyperMesh is executed. Figure 7.27 shows the import model menu. The housing origin is denoted by the green point.

2. Next, we create the 10 nodes where we will define the condensed stiffness. Figure 7.28 shows a screenshot of the 10 node locations. Within HyperMesh, selecting: Geometry > Nodes > XYZ brings up the node creation menu. The first 6 nodes are the interface nodes and are located at the shaft bearing midpoints. The 4 housing base nodes are constrained nodes and are located near the corners of the base as shown.

3. We use rigid body elements to connect each of the nodes created in (2) to the nearest surface nodes. Doing so connects the nodes to the housing model, allowing us to calculate the equivalent stiffness at the nodes. To create the elements in HyperMesh we select: 1D > rigids from the graphical interface at the bottom of the application window, bringing up the screen shown in Figure 7.29. To create the elements we select Create, check all dof checkboxes, and set elem_types = RBE2. We then select the one of the nodes created in (2) and the nodes to connect using RBE2 elements.
Figure 7.27: Importing the Nastran FE model and locating the origin.

Figure 7.28: Create interface nodes.
Figure 7.29: Creating RBE2 rigid body elements.

Figure 7.29 shows the created node (black) and the connecting nodes (white) for the first shaft hole interface. We select the nodes at the bearing outer race-housing interface as the connecting nodes to for the 6 interface nodes.

4. In order for RADIOSS to condense the housing mesh to the 10 nodes created in (2), we must identify them using the ASET load collector. We do so by selecting: Collectors > Create > Load Collectors from the dropdown menu. Doing so brings up the Create Load Collector menu as shown at the top of Figure 7.30. We set the collector name to 'ASET' but do not attach a card image. We then select the menu options shown in the bottom screen of Figure 7.30. Selecting each node one-by-one and clicking on the green create box applies the collector to each node. Note, these nodes must have 6 degrees of freedom for the mesh to condense properly.

5. We now create and define a load collector card for the solution method to be used for the condensing of the FE model. We create a load collector called CMSMETH by choosing Collectors > Create > Load Collectors as shown at the top of Figure 7.31. We choose CMSMETH as the card image for the load collector in the Create Load Collector window and click Create. This brings up the card definition user interface shown at the bottom of Figure
Figure 7.30: Creating and applying the ASET load collector.
7.31. We set the solution method to Guyan. This method is to condense a stiffness matrix for a static analysis. The Guyan method does not condense the mass matrix. The card must then be set within the Analysis > Control Cards > Global, Case, Control menu by checking the CMSMETH checkbox and CMSMETH = 3 (3 is the ID number of the CMSMETH card).

6. Finally, we must set the output parameters and file type we wish to generate. We do so within the Analysis > Control Cards > PARAM menu shown at the top of Figure 7.32. We check only the EXTOUT checkbox from the list of parameters and set EXTOUT = DMIGPCH to output a punch (.pch) file with the condensed matrices KAAX and MAAX. Radioss is executed from the graphical interface by selecting Analysis > Radioss and completing the fields as shown in the bottom of Figure 7.32. This result is a punch (.pch) file containing the condensed matrices KAAX and MAAX. The MAAX matrix is defined at the end of the output file as an empty matrix. Transmission3D does not recognize this, so the MAAX matrix must be deleted before proceeding.
Figure 7.32: Defining the output type and condensing the model.
### 7.4.2.2 The Condensed Housing Menus

The housing menu for the condensed housing matrix method is given in Figure 7.33. We now set TYPE = NASTRAN to import the Nastran bulk file of FILENAME = housing_nastrandata2_AX.pch. This file is located in the SAMPLES/ReductionSetWithHousing subdirectory under the default working directory. We set STIFFMATNAME = KAAX and MASSMATNAME = MAAX and NNODES = 10. The mass matrix must still be defined in Transmission3D even if it does not exist.

COORDXFM is the coordinate transformation required to align the global Transmission3D model reference frame with the housing reference frame. Using the FE housing method, the housing and global coordinate systems were aligned and coincident. Comparing Figure 7.23 and Figure 7.26, the transformation can be obtained. In the global coordinated system, the global reference frame must be translated by \( Y = 166 \) and \( X = -41 \). The global reference frame must then be rotated about its y-axis by -90 degrees in order to align it with the housing reference frame of Figure 7.26. Referring to the COORDXFM in Figure 7.33, the ‘Translate’ operation is listed second and performed first. The global reference frame is shifted along the y-axis (\( e_2 \)) by 166 and along the z-axis (\( e_3 \)) by -41. The ‘Rotate’ operation is performed second and rotates the translated global reference frame by \( -\frac{\pi}{2} \) about the y-axis.

The node menu is used to locate the 10 node locations of the condensed stiffness matrix. The node menus for nodes 1 and 5 are shown in Figures 7.34 and 7.35, respectively. For node 1, we enter IDNODE 3333. The node ID is embedded within the output file and can be obtained by numbering the 10 nodes within HyperMesh. The order does not matter, so long as the information in the node menu for each node is consistent. In this example, nodes 1-4 are the base nodes and nodes 5-10 are the nodes created at the shaft bearing origins. We check CONSTRAIN for the base surface nodes 1-4. XNODE, YNODE, and ZNODE are the node coordinates given in the housing coordinate system.

**Figure 7.33: The housing menu for the condensed housing model.**

<table>
<thead>
<tr>
<th>Feature</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>FILENAME</td>
<td>housing_nastrandata2_AX.pch</td>
</tr>
<tr>
<td>STIFFMATNAME</td>
<td>KAAX</td>
</tr>
<tr>
<td>MASSMATNAME</td>
<td>MAAX</td>
</tr>
<tr>
<td>NNODES</td>
<td>10</td>
</tr>
<tr>
<td>COORDXFM</td>
<td>Rotate(( -\frac{\pi}{2}))( e_2 ))Translate(166( e_2 ))( -41( e_3 ))</td>
</tr>
</tbody>
</table>
Figure 7.34: Node menu for node 1.

Figure 7.35: Node menu for node 5.
HELICAL REDUCTION SYSTEM WITH HOUSING

Table 7.1: The connector origin and race positions.

<table>
<thead>
<tr>
<th>Connector</th>
<th>XPOS</th>
<th>YPOS</th>
<th>ZPOS</th>
<th>AXPOS1</th>
<th>AXPOS2</th>
<th>DIA</th>
<th>AXPOS1</th>
<th>AXPOS2</th>
<th>DIA</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0.00</td>
<td>0.00</td>
<td>-37.50</td>
<td>-7.50</td>
<td>7.50</td>
<td>20.00</td>
<td>-7.50</td>
<td>7.50</td>
<td>47.00</td>
</tr>
<tr>
<td>2</td>
<td>0.00</td>
<td>72.00</td>
<td>82.50</td>
<td>-7.50</td>
<td>7.50</td>
<td>20.00</td>
<td>-7.50</td>
<td>7.50</td>
<td>47.00</td>
</tr>
<tr>
<td>3</td>
<td>0.00</td>
<td>72.00</td>
<td>-38.50</td>
<td>-8.50</td>
<td>8.50</td>
<td>30.00</td>
<td>-8.50</td>
<td>8.50</td>
<td>62.00</td>
</tr>
<tr>
<td>4</td>
<td>0.00</td>
<td>72.00</td>
<td>83.50</td>
<td>-8.50</td>
<td>8.50</td>
<td>30.00</td>
<td>-8.50</td>
<td>8.50</td>
<td>62.00</td>
</tr>
<tr>
<td>5</td>
<td>0.00</td>
<td>166.00</td>
<td>-41.00</td>
<td>-11.00</td>
<td>11.00</td>
<td>50.00</td>
<td>-11.00</td>
<td>11.00</td>
<td>90.00</td>
</tr>
<tr>
<td>6</td>
<td>0.00</td>
<td>166.00</td>
<td>86.00</td>
<td>-11.00</td>
<td>11.00</td>
<td>50.00</td>
<td>-11.00</td>
<td>11.00</td>
<td>90.00</td>
</tr>
</tbody>
</table>

7.5 Connectors

The reduction model consists of 6 connectors. We discuss how to model these using both stiffness and tapered roller bearings in this section. We also discuss how we can model shaft and housing misalignment by introducing the undeformed deformation and runout error inputs within the connectors menu. We begin by discussing stiffness bearings and follow with the discussion of the tapered roller bearings. In order to model the 6 connectors, we first set ENABLECONNECTORS = TRUE, NCONNECTORS = 6 within the edit menu.

7.5.1 Stiffness Bearings

Figure 7.36 shows the connector menu for the stiffness bearing connector 1. We define the position of the bearing origin in the global coordinate system within the connector menu. The bearing origin must coincide with the corresponding interface node of the condensed housing stiffness matrix so we need to transform the housing node coordinates to the global reference frame. Referring back to Figure 7.26, the coordinates in the housing reference frame of the first input shaft node are XPOS = 3.5, YPOS = -166, and ZPOS = 0. To convert these coordinates to the global reference frame, we simply apply the reverse of the transformation used in Section 7.4.2.2. The new transformation becomes: Translate(166 ∗ e2 + -41 ∗ e3) ∗ Rotate(-pi/2 ∗ e2). First, we apply the rotation of -90 degrees about the y-axis to align the two reference frames. After this intermediate step our node coordinates become: XPOS = 0, YPOS = -166, ZPOS = -3.5. Now, we apply the translation to position the housing reference frame origin coincident with the global origin and obtain the node coordinates in the global reference frame. The node coordinates after translation become: XPOS = 0, YPOS = 0, ZPOS = -37.5.

Figure 7.36 shows the connector menu for connector 1 with the bearing origin coordinates XPOS = 0, YPOS = 0, ZPOS = -37.5. Applying the same transformation on the remaining interface nodes from the node submenu within the housing menu results in the exact bearing origin coordinates required to match those of the condensed housing stiffness matrix. The resulting bearing origin coordinates for each of the 6 bearings are provided in Table 11.2.

The race dimensions are also entered into the connector menu. Table 11.2 shows the race dimensions for each of the 6 bearings. KR and KZ are the bearing stiffness values in the radial and axial directions. For each bearing in this model we use KR = KZ = 1E+06. KTHETAR and KTHETAZ are the rotational stiffnesses about the radial and axial axis. We use values of KTHETAR = 1E+08 and KTHETAZ = 0 for each bearing.

We use the unloaded deformation feature to model misalignment of the input shaft by checking the UNLOADED-DEFM checkbox. UX, UY, UZ and THETAX, THETAY, THETA are the linear and angular displacement of the origin of the inner bearing race. In this example we shift the inner race of connector 1 in the negative x-direction by entering UX = -0.524 as shown in 7.36. Similarly, we shift the inner race of bearing 2 in the positive z-direction by setting UX = 0.524 in the connector 2 menu. Shifting both bearings misaligns the input shaft by an angle of 0.5 degrees with the y-z plane.
7.5.2 Roller Bearings

Alternatively, we can model the 6 connectors using tapered roller bearings. Doing so removes the error associated with the approximated stiffness values of the stiffness type connectors and allows us to model the roller element contact. Figure 7.38 shows the connector menu for the roller bearing connector 2. The bearing origin and race locations for the tapered roller bearings are the same as those given in Table 11.2 for the stiffness bearings.

Selecting TYPE = ROLLERS enables the roller bearing submenus as shown in Figure 7.38. The geometry menu contains the inputs that describe the roller geometry. Figures 7.37 and 7.39 show the geometry menu input parameters for connector 2. The geometry menu inputs for the remaining tapered roller connectors are provided in Table 7.2.

Within the cage menu, we check the AUTOCOMPUTE checkbox to calculate the cage stiffness based upon the geometry of the rollers and races. The contact grid is used to define the contact grid on the roller surfaces. SEPTOL is the maximum separation distance between two contacting surfaces that is used to define a contact pair. We set SEPTOL to 0.6 for connectors 1 and 2 since the input shaft bearing races have been displaced. Connectors 3-6 have SEPTOL = 0.1. NPROFIVS is the number of contact grid cells on each side of the center cell in the profile direction. We choose NPROFIVS = 1 so the total number of grid cells in the profile direction is $2 \times NPROFIVS + 1 = 3$. DSPOF sets the size of each cell in the profile direction. We choose DSPOF = 0.2 in this example. NFACEDIVS = 3 sets the number of grid cells on each side of the center cell in the facewidth direction to 3. The FE model menu is used to select the type and number of elements on the roller element. We set ELEMTYPE = QUADRATIC, NCIRCDIVS = 32, and NCIRCDIVS2 = 16. The material properties are entered into the material menu.

The runout submenu for roller connectors is similar to the unloaded deformation discussed previously for stiffness bearings. Within the runout menu we can move the bearing inner race (race1) or the outer race (race2). Moving the inner race simulates shaft misalignment in the same way as the unloaded deformation. Moving the outer race simulates housing manufacturing errors. For connector 2, we shift the outer race by MAGRUNOUT2 = 0.524 in the positive x-direction (ANGRUNOUT2 = 0). ANGRUNOUT is the angle of the displacement direction and follows the right-hand rule about the bearing rotational axis. Notice, using the runout error we cannot rotate the bearing race as we could have using the unloaded deformation feature with stiffness bearings. We do have the option to displace either the inner or outer race, or both, using the runout error. With stiffness connectors we only have the option to move the inner race.
Figure 7.37: Roller bearing connector 2 details.

Figure 7.38: Connector menu for roller bearing connector 2.

Table 7.2: The connector roller geometry parameters.

<table>
<thead>
<tr>
<th>Connector</th>
<th>NROLLERS</th>
<th>PITCH DIA</th>
<th>CONTACT ANGLE</th>
<th>LENGTH</th>
<th>RAD CLEAR</th>
<th>ROLLER DIA</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>18</td>
<td>33.50</td>
<td>14.00</td>
<td>10.00</td>
<td>0.001</td>
<td>5.00</td>
</tr>
<tr>
<td>2</td>
<td>18</td>
<td>33.50</td>
<td>14.00</td>
<td>10.00</td>
<td>0.001</td>
<td>5.00</td>
</tr>
<tr>
<td>3</td>
<td>20</td>
<td>46.00</td>
<td>-14.00</td>
<td>12.00</td>
<td>0.001</td>
<td>6.00</td>
</tr>
<tr>
<td>4</td>
<td>20</td>
<td>46.00</td>
<td>14.00</td>
<td>12.00</td>
<td>0.001</td>
<td>6.00</td>
</tr>
<tr>
<td>5</td>
<td>24</td>
<td>70.00</td>
<td>-14.00</td>
<td>14.00</td>
<td>0.001</td>
<td>8.00</td>
</tr>
<tr>
<td>6</td>
<td>24</td>
<td>70.00</td>
<td>14.00</td>
<td>14.00</td>
<td>0.001</td>
<td>8.00</td>
</tr>
</tbody>
</table>
Table 7.3: The roller bearing runout error.

<table>
<thead>
<tr>
<th>Connector</th>
<th>MAGRUN OUT1</th>
<th>ANGRUN OUT1</th>
<th>MAGRUN OUT2</th>
<th>ANGRUN OUT2</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0</td>
<td>0</td>
<td>0.524</td>
<td>0</td>
</tr>
<tr>
<td>2</td>
<td>0</td>
<td>0</td>
<td>0.524</td>
<td>180</td>
</tr>
</tbody>
</table>

Figure 7.39: Connector geometry menu for roller bearing connector 2.
The helical reduction system consists of two contact pairs. Contact pairs are modeled within the pairs menu, shown in Figure 7.40 for pair number 1. The first pair connects the input rotor sun 1 to the idler rotor sun 1. We define the two contacting gears by setting TYPE = SUN_SUN and selecting IROTOR1 = 1, ISUN1 = 1 and IROTOR2 = 2, ISUN2 = 1. SEPTOL, NPROFDIVS, DSPROF, and NFACEDIVS are defined in the same way here as they were in the connectors grid menu. We check the SAMEGRIDPARAMS checkbox to apply the same grid parameters to each gear pair.

### Figure 7.40: The pairs menu.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>SEPTOL</td>
<td>0.1000000000</td>
</tr>
<tr>
<td>NPROFDIVS</td>
<td>3</td>
</tr>
<tr>
<td>DSPROF</td>
<td>0.0200000000</td>
</tr>
<tr>
<td>NFACEDIVS</td>
<td>3</td>
</tr>
<tr>
<td>MU</td>
<td>0.0000000000e+00</td>
</tr>
<tr>
<td>BACKCONTACT</td>
<td></td>
</tr>
<tr>
<td>NPAIRS</td>
<td>2</td>
</tr>
<tr>
<td>PAIR</td>
<td>1</td>
</tr>
<tr>
<td>TYPE</td>
<td>SUN_SUN</td>
</tr>
<tr>
<td>IROTOR1</td>
<td>1</td>
</tr>
<tr>
<td>ISUN1</td>
<td>1</td>
</tr>
<tr>
<td>IROTOR2</td>
<td>2</td>
</tr>
<tr>
<td>ISUN2</td>
<td>1</td>
</tr>
</tbody>
</table>

### 7.6 Pairs

The helical reduction system consists of two contact pairs. Contact pairs are modeled within the pairs menu, shown in Figure 7.40 for pair number 1. The first pair connects the input rotor sun 1 to the idler rotor sun 1. We define the two contacting gears by setting TYPE = SUN_SUN and selecting IROTOR1 = 1, ISUN1 = 1 and IROTOR2 = 2, ISUN2 = 1. SEPTOL, NPROFDIVS, DSPROF, and NFACEDIVS are defined in the same way here as they were in the connectors grid menu. We check the SAMEGRIDPARAMS checkbox to apply the same grid parameters to each gear pair.
7.7 Analysis Setup

We set up the analysis parameters in the setup and range menus. The two menus are provided in Figures 7.41 and 7.42. Within the setup menu, we set the INITIALTIME = 0.00. We write the analysis results to the postprocessing file postproc.dat. We split the postprocessing file into separate files for each individual time step by checking the SPLITPOSTPROCFILE checkbox.

We use the range menu to set the cycle parameters of the analysis. The mesh cycle time for the InputSun1-IdlerSun1 gear pair is calculated to be 1.00001 seconds. We run the analysis for NTIMESTEPS = 11 so we divide the cycle time by 10 to get DELTAT = 0.100001.

7.8 Analysis Results

The contact pressure distribution of the input rotor gear is shown below for the following three cases: stiffness bearing model without misalignment, stiffness bearing model with shaft misalignment, and roller bearing model with housing hole misalignment. The contact pattern shifts to one side of the gear in the cases where misalignment is present. The resulting distribution is very similar for the two misalignment cases, though the pressure does slightly differ.
Figure 7.42: The analysis range menu.
Figure 7.43: The contact pressure on the input gear for the three misalignment cases.
Quality measurements of large gears after manufacturing can become quite difficult due to the deflections associated with the weight of such gears. Transmission3D provides a computational means to correct for these errors, which is the subject of the example in this chapter.

We describe how to set up the simple model shown in Figure 8.1. We also detail the process for generating lead and profile charts of the gear tooth deflection, which can then be used to correct for the measurement errors. The session file GravitationalDeflectionRingGear.ses is located in the SAMPLES/GravitationalDeflectionRingGear within the default directory. We use the Newton as the unit of force, meter for length, and second for time in this example.

The model consists of 4 rotors so we set NROTORS = 4 within the edit menu. We define contact between the ring gear rim and the three support shafts, so ENABLEPAIRS is also checked. Selecting ENABLEBODYFORCE allows us to include the weight of the model components by entering the gravitational acceleration as BODYFORCE = -9.81.

The Ring Gear rotor is the first rotor and is comprised of a ring gear and a shaft. Each of the remaining 3 rotors are the support shafts used to balance the body force of the ring gear rotor. The support shafts are each modeled with a single-segment shaft. We wish to keep each of the above rotors stationary so we set each TYPE = INPUT with RPM = 0.
Figure 8.1: The ring gear model.
8.1 The Ring Rotor

The ring rotor shown in Figure 8.1 consists of a ring gear and a shaft. The rotor origin is coincident with the global origin (XPOS = 0, YPOS = 0, ZPOS = 0) with the rotational axis set to the positive z-axis (AX = 0, AY = 0, AZ = 1). The rotor motion is confined to linear motion along the z-axis, the same direction as the force exerted from the weight of the gear. To do this we set UXCONSTRAINT = UYCONSTRAINT = THETAXCONSTRAINT = THETAYCONSTRAINT = TRUE and TYPE = INPUT, RPM = 0. The UZCONSTRAINT box is left unchecked. The rotor 1 menu is provided in Figure 8.2.

The ring gear is coincident with the rotor origin (AXIALPOSNSHAFT = 0). The tooth menu for the gear is shown in Figure 8.3. Within the base menu we set CIRCORDER = 128 and AXIALORDER = 4.

The ring gear rim is modeled using a single-segment shaft. The shaft input parameters are entered into the shaft menu of rotor 2. Setting AXIALPOSNSHAFT = -0.15 shifts the starting point of the shaft by half of the ring gear face width in the negative z-direction. The segment menu inputs are provided in Figure 8.4. The shaft INNERDIA = 2.1 to match the outer diameter of the ring gear base surface. We set IDRACE = TRUE, CIRCORDER = 128, and AXIALORDER = 4 in order to mate the inner surface of the shaft to the outside diametral surface of the ring. ODCONSTRAINED = TRUE, ODTYPE = FLEXIBLE sets the shaft constraint on the rotor to flexible on the outside diameter of the shaft. Doing so properly constrains the rotor while also allowing the shaft to deform since we are interested in the deflection of the ring gear. Finally, since we left the UZCONSTRAINT box unchecked for the rotor, we must constrain this component using a contact constraint. We do so by checking the BACKCONTACT box to allow the back side of the shaft to make contact with the 3 support shafts shown in Figure 8.1.
<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Tooth Number</td>
<td>150</td>
</tr>
<tr>
<td>Number of Face Elements</td>
<td>4</td>
</tr>
<tr>
<td>Cog Order</td>
<td>10</td>
</tr>
<tr>
<td>Prop Type</td>
<td>SIMPLE</td>
</tr>
<tr>
<td>Symmetry</td>
<td>NORMAL</td>
</tr>
<tr>
<td>Plane</td>
<td>NORMAL</td>
</tr>
<tr>
<td>Normal Module</td>
<td>0.125370e+0</td>
</tr>
<tr>
<td>Normal Press Angle</td>
<td>22.500000e+0</td>
</tr>
<tr>
<td>Normal Thickness</td>
<td>0.020300e+0</td>
</tr>
<tr>
<td>Face Width</td>
<td>0.001000e+0</td>
</tr>
</tbody>
</table>

**Figure 8.3: The ring gear tooth menu.**

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Segment</td>
<td>1</td>
</tr>
<tr>
<td>Type</td>
<td>DEFINE GEOMETRY</td>
</tr>
<tr>
<td>Length</td>
<td>0.300000e+0</td>
</tr>
<tr>
<td>Outer Shape</td>
<td>CYLINDRICAL</td>
</tr>
<tr>
<td>Outer Dia</td>
<td>2.108266e+0</td>
</tr>
<tr>
<td>Edge Radius</td>
<td>0.000000e+0</td>
</tr>
<tr>
<td>Edge Dia</td>
<td>0.000000e+0</td>
</tr>
<tr>
<td>Outer Curvature</td>
<td>0.000000e+0</td>
</tr>
<tr>
<td>Inner Shape</td>
<td>CYLINDRICAL</td>
</tr>
<tr>
<td>Inner Dia</td>
<td>2.108000e+0</td>
</tr>
<tr>
<td>Edge Radius</td>
<td>0.000000e+0</td>
</tr>
<tr>
<td>Edge Dia</td>
<td>0.000000e+0</td>
</tr>
</tbody>
</table>

**Figure 8.4: The ring gear shaft segment menu.**
8.2 The Support Shaft Rotors

The three support shaft rotors shown in Figure 8.1 are used to constrain the axial motion of the ring gear. This simulates a support structure which might be used in a manufacturing environment to measure the gear parameters. Each shaft is modeled using a single-segment partial shaft. The rotor menu for support shaft 1 is given in Figure 8.5.

Since the support shafts are partial shafts with the same radius of curvature as the ring gear, the rotor origin of each shaft must coincide with the ring gear shaft origin (XPOS = YPOS = ZPOS = 0). We set the positive rotational axis for each support shaft to the negative z-axis (AX = AY = 0, AZ = -1). We constrain the rotational motion about the z-axis axis by setting TYPE = INPUT and RPM = 0. Each of the support shafts’ remaining degrees of freedom are constrained by setting UXCONSTRAINT = UYCONSTRAINT = UZCONSTRAINT = TRUE.

The segment menu for support shaft 1 is shown in Figure 8.6. We set OUTERDIA = 2.109266 to match the outside diameter of the ring gear shaft. INNERDIA = 2.03 sets the inside diameter of the shaft just outside of the root of the gear. We model the support shafts with partial shafts by checking the PARTIAL checkbox. Doing so enables the THETABEGIN and THETAEND input fields. Table 8.1 provides the beginning and ending angles used for each of the 3 support shafts.

The angles were chosen by first setting up a model with 3 equally spaced, fully cylindrical support shafts. A preprocessing iGlass file was generated to view the FE mesh at the ring gear shaft–support shaft interface. We then selected partial shaft angles that coincided with the ring gear shaft element edges at equally spaced distances from the support shaft center points.

We select the ODCONSTRAINED checkbox for each of the 3 shafts and ODTYPE = RIGID. This constrains the nodes of the outside diametral surfaces of the support shafts to their respective rotor reference frames in order to keep their stiffness matrices non-singular. BACKCONTACT is also checked in order to define the back of each shaft as a contact surface.
Table 8.1: Start and end angles for partial support shafts.

<table>
<thead>
<tr>
<th>Rotor Number</th>
<th>THETA BEGIN</th>
<th>THETA END</th>
</tr>
</thead>
<tbody>
<tr>
<td>2</td>
<td>53.4375</td>
<td>64.6876</td>
</tr>
<tr>
<td>3</td>
<td>174.3750</td>
<td>185.6250</td>
</tr>
<tr>
<td>4</td>
<td>295.3125</td>
<td>306.5625</td>
</tr>
</tbody>
</table>

Figure 8.6: The shaft segment menu for shaft support 1.
8.3 The Pairs Menu

We define the contact between the back of the ring gear shaft and each of the three shaft supports within the pairs menu. These contacts constrain the axial motion of the ring gear and apply reaction forces on the ring gear shaft to oppose the weight of the gear. The pairs menu for the pair between the ring gear shaft and support shaft 1 is provided in Figure 8.7. Within this menu, we set the contact TYPE = SHAFT_SHAFT. NDIVS1 and NDIVS2 set the number of divisions per element in the circular and radial directions to use for contact points. In this case, we set each to 1. The first contact surface is identified by setting IROTOR1 = 1, ISHAFT1 = 1, ISEG1 = 1, and SEGSURF1 = BACK. The support shaft contact surface is identified by setting IROTOR2 = 2, ISHAFT2 = 1, ISEG2 = 1, and SEGSURF2 = BACK.

8.4 Generating the Lead and Profile Charts

The analysis is set up using the setup and range menus. In this example, we run the analysis for NTIMESTEPS = 1. The cycle time a DELTATIME are not of importance in this example since we are no mesh cycle is being analyzed. The results are written to the postprocessing file postproc.dat.

We have written two calyx script files to generate lead and profile charts of the normal tooth deviation. The lead and profile chart script files, LeadChartScript.txt and ProfileChartScript.txt, are located in the SAMPLES /GravitationalDeflectionRingGear subdirectory within the default directory. The lead chart script generates a table of the normal tooth deviation for a given roll angle along the facewidth of the tooth. The profile chart script generates a table of the normal tooth deviation for a given face location along the profile of the tooth.
These scrips use the information contained within the calyxtmp/system.cfg and postproc.dat files, along with user defined inputs, to output the lead and profile data. The scripts are written such that the user may enter the number of data points desired. The data points represent the face locations for the lead chart. In the lead chart script, the roll angle is also required as an input. We then identify the rotor, ring, side, and body number for the tooth side of interest. The rotor and ring number are determined in the Guide menus, while the body number is output to the information window when generating a preprocessing IGlass file. The tooth side number can be determine by using the right-hand rule. With the thumb pointing in the direction of the positive rotor axis, side 1 is the side which the finger tips reach just before the tip of the gear tooth. Figure 8.8 provides a visual explanation of this definition. The final user input required for the lead chart script is the postprocessing filename postproc.dat.

The program first calculates the base radius at the roll angle entered by the user to ensure it falls within the tooth profile region. Next, an iterative calculation is performed to locate the s-coordinate at the given roll angle since calyx operates in the (s,t) coordinate system. The postprocessing file is then read for the body handle, surface name, tooth instance, and s data for each t coordinate (defined by the number of data points in the facewidth direction). A data file titled LeadChart.dat is the output of the program and contains the roll angle used, surface name, and lead table with normal deviations in millimeters.

The profile chart program inputs are: face location, starting roll angle, number of data points in the profile direction and rotor, ring, side, and body numbers. The same postprocessing output file, postproc.dat, is entered in the input variables section at the top of the script. The profile chart program also calculates the base radius in order to find the roll angle at the tip of the gear tooth. The function GetSurfaceParameterfromRollAngle converts a given roll angle into a corresponding s coordinate. The t coordinate is calculated based upon the face location entered by the user in the inputs section. The normal deviation data is then retrieved from the calyxtmp/system.cfg file for the desired body handle, surface name, tooth instance number, s, and t and written to the output file ProfileChart.dat. The s coordinate is determined based upon the start roll angle, tip roll angle, and number of data points selected. The GetSurfaceParameterfromRollAngle function is called for each roll angle iteration to convert the roll angle into an s coordinate which can then be used to retrieve the deviation data at the desired location.
CHAPTER 9

SIMPLE FE CARRIER PLANETARY SYSTEM

The simple FE carrier planetary model presented in this chapter is pictured in Figure 9.1. The model consists of 3 rotors: the Sun Rotor, Ring Rotor, and Carrier Rotor. The focus of this example is on the carrier rotor, in particular, and the process used to import an FE carrier into Transmission3D using the Guide menu inputs.

The inputs for this model can be loaded using the session file FECARRIER_STIFFBRG.ses located within the SAMPLES/PlanetaryExamples/FECarrier subdirectory. The units used throughout this example are Metric Engineering (N, mm, s) units.

The model consists of three rotors, namely the Sun Rotor, Ring Rotor, and Carrier Rotor. The sun rotor is set to TYPE = INPUT with a RPM = 1000. We would like to hold the ring rotor stationary so we set it to TYPE = INPUT with and RPM = 0. The carrier rotor is set to TYPE = OUTPUT with TORQUE = 4E+05.

The model also contains connectors so we select ENABLECONNECTORS and set NCONNECTORS = 2 within the EDIT menu. We also select ENABLEPAIRS in order to enable the definition of gear contacts.
Figure 9.1: The simple FE carrier planetary model.
9.1 The Sun Rotor

The sun rotor consists of a sun gear and a shaft. The sun rotor menu is shown in Figure 9.2. The sun rotor origin is coincident with the rotor origin so we set XPOSN, YPOSN, and ZPOSN to 0. The global z-axis is selected as the rotor rotational axis by entering AX = AY = 0, and AY = 1. Positive rotation of the sun rotor follows the right hand rule about this axis. Selecting ENABLESHAFTS and ENABLESUNS activates the shaft and sun submenus. We set the number of shafts and suns each to 1.

The sun rotor is constrained by two bearings presented in the Connectors section so we leave all degrees of freedom of the rotor's reference frame unconstrained. This is done by setting UXCONSTRAINT, UYCONSTRAINT, UZCONSTRAINT, THETAXCONSTRAINT, and THETAYCONSTRAINT to FALSE.

9.1.1 Shaft

The sun rotor shaft is modeled within the SHAFT submenu. Figure 9.3 shows the shaft menu inputs. The beginning of the shaft coincides with the rotor origin so we set AXIALPOSNSHAFT = 0. The material properties shown are standard for steel and are consistent for all components in this example.

The shaft segment details are shown in Figure 9.4 and are entered into the SEGMENT submenu. The outer diametral surface of segment 1 is constrained to the rotor reference using a FLEXIBLE constraint type.

9.1.2 Sun

The sun menu of the sun rotor is pictured in Figure 9.5. The mid-face of the sun is positioned at a distance of AXIALPOSN = 75.00 from the rotor origin. The Fourier series coefficients used to interpolate between elements at the base of the gear are AXIALORDER = 2 and CIRCORDER = 4 within the BASE menu. The sun gear TOOTH menu with the gear tooth input parameters is shown in Figure 9.6.
Figure 9.3: The sun rotor shaft menu.

Figure 9.4: The sun rotor shaft details.
Figure 9.5: The sun rotor sun menu.

Figure 9.6: The sun rotor sun tooth menu.
9.2 The Ring Rotor

The ring rotor is made up of a single segment shaft and a ring gear. The ring rotor menu is pictured in Figure 9.7. We set the rotor origin at XPOSN = 0, YPOS = 0, ZPOS = 60 and the rotational axis to AX = AY = 0, AZ = 1. We fully constrain the ring rotor reference frame since no connectors are attached to the rotor.

9.2.1 Shaft

The ring rotor shaft menu is shown in Figure 9.8. The beginning of the shaft coincides with the rotor origin, so AXIALPOSNSHAFT is set to 0. The shaft acts as a rim for the internal ring gear, so only one segment is needed. The inner diametral surface of the shaft contains a race for the ring gear connection, while the outer surface nodes are constrained the rotor reference frame using a RIGID shaft constraint. The shaft segment details are shown in Figure 9.9.

9.2.2 Ring

The ring gear is modeled within the RING menu shown in Figure 9.10. The ring gear mid-face is positioned at an AXIALPOSN = 15.00 from the rotor origin. The ring gear tooth input parameters are entered within the TOOTH menu shown in Figure 9.11.

Figure 9.7: The ring rotor menu.
Figure 9.8: The ring rotor shaft menu.
Figure 9.9: The ring rotor shaft details.
Figure 9.10: The ring rotor ring menu.

Figure 9.11: The ring rotor ring tooth menu.
9.3 The Carrier Rotor

The carrier rotor consists of a carrier and a shaft. Figure 9.12 shows the rotor menu for the carrier rotor. The rotor origin is positioned within the global reference frame at XPOSN = YPOSN = 0, ZPOS = 75. We select ENABLECARRIERS and ENABLESHAFTS and set NCARRIERS = NSHAFTS = 1. The rotor is connected to ground through a bearing, so we leave UXCONSTRAINT, UYCONSTRAINT, UZCONSTRAINT, THETA XCONSTRAINT, and THETA YCONSTRAINT unchecked.

9.3.1 Shaft

The shaft menu for the carrier shaft is shown in Figure 9.13. The shaft starts at an AXIALPOSNSHAFT = 20 and has NSEGMENTS = 4. The shaft’s segment details are provided in Figure 9.14. The first segment connects to the carrier race, while the fourth segment contains the shaft constraint for the rotor. The constraint is of the FLEXIBLE type.

9.3.2 Carrier

The carrier is modeled within the CARRIER menu shown in Figure 9.15. The carrier details are also provided in Figure 9.16. We would like to import a finite element carrier model in the Nastran (.bdf) file format, so we set the TYPE to FECARRIER_NASTRAN. The carrier in this example is contained within one file, so we set NFECARRIERFILES = 1. REMOVEOPTIONALNODES allows the user to remove the mid-nodes of quadratic elements. In this example we elect not to do so and leave this box unselected. The NODETOLERANCE input field is set to 0.001, which allows 0.001 length unit of error when searching for nodes at connecting (race) surfaces. The MAXJOINTANGLE input sets the maximum angle for smoothing sharp edges of joining elements. In this case, we set the angle to the recommended 15 degrees. The carrier origin is coincident with the rotor origin, so AXIALSHIFT is set to 0. The carrier contains 4 pinion gears at equally spaced interval. In Transmission3D, identical pinion gears are counted only once, so we set NPINIONS = 1. NGROUPS is set to 4, since there are 4 copies of the pinion. The carrier requires one connecting surface in order to connect the output shaft so we set NRACES = 1.
Figure 9.13: The carrier rotor shaft menu.

Figure 9.14: The carrier rotor shaft details.
The FILE submenu is used to enter the information about the Nastran file. The NASTRANFILENAME is CarrierData.bdf. This file is also located within the SAMPLES/ PlanetaryExamples/ FECarrier subdirectory. The preferred cutting direction is chosen to be the 'Z' axis in this example. It is always most efficient to select the rotational axis as the cutting direction for computation time purposes.

The RACE submenu is where the information regarding the connecting surfaces is entered. The race details are also shown in Figure 9.16. The race TYPE is set to CYLINDRICAL, and DIAMETER = 60. AXPOSN1 and AXPOSN2 set the axial distances from the carrier origin to the front and back edges of the race, respectively. We set AXPOSN1 = 20 and AXPOSN2 = 40. CIRCORDER = 8 and AXIALORDER = 1 must match the Fourier series orders of interpolation between elements at the mating shaft surface.

### 9.3.2.1 Carrier Pinion

The carrier pinion is modeled within the PINION submenu shown in Figure 9.17. The pinion in this example is a HELICAL pinion and it is located at a RADPOSN = 70. We suggest using the COMPOUND pin and pinion shaft types as we have done here. AXIALCONSTRAINT = TRUE constrains the pinion from motion in the axial direction since we do not wish to model contact between the pinion and carrier.

The PINHOLES menu (Figure 9.18) is used to define the race surfaces on the carrier where the pinion pin connects. The pin in this example is supported at both ends so we must define the position and radius of the hole at the front and the back side. To do so, set TYPE_HOLE = COMPOUND and NPINHOLES = 2 in the PINION menu. Within the PINHOLES menu, the back (Z1) and front (Z2) sides of the hole are defined, along with the diameter of the hole. The AXIALORDER and CIRCORDER parameters in the PINHOLES menu must match the orders used at the corresponding segments of the pin shaft.

### 9.3.2.2 Pin Shaft

The carrier pin shaft is modeled in the PINSHAFT menu. The pin shaft details are shown in Figure 9.19. The first shaft segment begins at AXIALPONSHAFT = -30. The first and third shaft segments connect to the carrier hole races defined in the CARRIERHOLES menu, while the middle segment connects to the inside diameter of the pinion bearing.
Figure 9.16: The FE carrier details.

Figure 9.17: The carrier pinion menu.
Figure 9.18: The carrier PINHOLES menu.

Figure 9.19: The carrier pin shaft details.
9.3.2.3 Pinion Bearing  The pinion bearing connects the pin shaft to the pinion shaft and is modeled within the BEARING submenu shown in Figure 9.20. AXIALPOS = 0 positions the bearing origin at the same axial position as the carrier origin. DIARACE2 is the diameter of the inner race that connects to the pin shaft, while DIARACE1 is the diameter at the outer race-pinion shaft connection. AXPOS1RACE1 and AXPOS2RACE1 set the axial positions of the outer race edges from the bearing origin. AXPOS1RACE2 and AXPOS2RACE2 do the same for the inner race. Selecting STIFFNESS for the bearing TYPE and setting STANDARD = TRUE allows us to model the bearing using a radial, axial, and bending spring stiffness value. These values are shown in the BEARING menu as KR, KZ, and KTHETA.

The bearing can also be modeled as a roller type bearing by selecting ROLLERS from the TYPE drop down input within the BEARING menu. The file FECARRIERROLLERBRG.ses is located within the same working directory as the FECARRIERSTIFFNESSBRG.ses model. The race positions remain the same as for the stiffness bearings and the roller bearing design parameters are entered within the GEOMETRY menu shown in Figure 9.21. The CAGE is set to AUTOCOMPUTE within the CAGE submenu. SEPTOL = 0.01, NPROFDIVS = 0, DSPROF = 0.1, and NFACEDIVS = 3 are entered into the CONTACT_GRID submenu, and the material properties for steel used up to this point are again used in the MATERIAL menu.

9.3.2.4 Pinion Shaft  The pinion shaft connects the outer race of the pinion bearing to the inside diameter of the pinion. The pinion shaft is modeled within the PINIONSHAFT submenu. The shaft consists of two segments with the first segment beginning at AXIALPOSNSHAFT = -15 from the carrier origin. The pinion shaft details are provided in Figure 9.22. The first shaft segment connects to the bearing at its inside diametral surface and to the inside diameter of the pinion at its outer surface. The second shaft segment is a very small segment and exists for the purpose of constraining each pinion. The FLEXIBLE constraint type is used for the pinion shaft constraint.

9.4 Deck  The DECK menu shown in Figure 9.23 is used to enter the inputs related to the pinion tooth. Here, we set the Fourier orders for the base surface of the pinion that connects to the pinion shaft. The TOOTH menu (Figure 9.24) is used to enter the inputs related to the tooth geometry.
Figure 9.21: The carrier pinion roller bearing geometry menu.

Figure 9.22: The carrier pinion shaft details.
Figure 9.23: The carrier pinion deck menu.
Figure 9.24: The carrier pinion tooth menu.
9.5 Connectors

Two connectors are used in this example, one on the carrier rotor output shaft and one on the sun rotor shaft. The connectors are modeled as stiffness bearings in the FECARRIER_STIFFNESSBRG.ses model and as roller bearings in the FECARRIER_ROLLERBRG.ses model. The connector menu inputs for the stiffness bearings are provided for the sun and carrier bearings in Figures 9.25 and 9.27, respectively.

To model the connectors as stiffness bearings, the TYPE is changed to ROLLERS in the CONNECTOR menu. The roller geometry inputs are entered within the geometry menus shown in Figures 9.28 and 9.28 for connectors 1 and 2, respectively.

9.6 Pairs

The PAIRS menu is used to define the gear contact pairs within the model. This particular example requires the use of two pairs, a sun-pinion pair and a pinion-ring pair. The pairs menu for the sun-pinion pair is shown in Figure 9.29. The connecting gears are defined by entering the indices which define each gear. In the case of the sun gear, we
Figure 9.27: The carrier bearing connector menu.

Figure 9.28: The carrier roller bearing geometry menu.
Figure 9.29: The pairs menu.

enter IROTOR = 1, ISUN = 1. The pinion gear is defined by entering: IROTOR = 3, ICARRIER = 1, IPINION = 1, IDECK = 1. The contact pair SEPTOL sets the maximum separation tolerance used to consider two surfaces as contact surfaces. NPROFDIVS and NFACEDIVS sets the number of contact grid division on each side of the mid-point in the profile and face directions, respectively. Setting ADAPTIVEGRID = TRUE automatically determines the correct grid spacing for the selected number of grid spaces in the profile direction.
CHAPTER 10

ESTIMATING MISALIGNMENT & LEAD CORRECTION OF HELICAL GEAR PAIR

This example demonstrates the use of Transmission3D to estimate the misalignment of helical gear pair due to the deflections in the system and calculating a lead correction needed to center the contact pattern.

10.1 Model

This is a simple gear pair system with two rotors. The rotor 1 consists of gear and long shaft, connected to the ground by two stiffness bearing. The rotor 2 consists gear which is held rigidly at the inside diameter.

10.2 Estimating Misalignment

The Transmission3D model is analyzed for one time step to get the load distribution on the gears along the face width as shown in Figure 10.5. The load distribution shows a shift in pattern towards an end of the face width. This is due to the overall deflection in the system. Transmission3D estimates the misalignment of helical mesh at every time of the analysis using the tooth and body reference frame deflections. The misalignment is calculated about the normal to line/plane of action in radians. The sign convention for the misalignment, positive misalignment shifts contact to \( \zeta = +1 \) end of the gear and negative misalignment shifts to \( \zeta = -1 \) end of the gear. In the case where the two gears axis’ are opposite, the +/- \( \zeta \) directions of member 1 are used to define positive/negative misalignment, respectively. Figures 10.2 and 10.3 graphically displays the definition of the gear pair misalignment. The outputs are saved to a file HELICALMISALIGNMENT.DAT inside the calyxtmp/ folder during the analysis. The format of the file is

\[
\begin{align*}
\text{Time} & \quad \theta_1 & \quad l_{s1} & \quad \theta_2 & \quad l_{s2} & \quad \theta_{31} & \quad l_{s31} & \quad \theta_{32} & \quad l_{s32} & \quad \theta_{33} & \quad l_{s33} & \quad \ldots
\end{align*}
\]
Figure 10.1: Helical Gear Pair Model.
Table 10.1: Gear Data

<table>
<thead>
<tr>
<th></th>
<th>Gear1</th>
<th>Gear2</th>
</tr>
</thead>
<tbody>
<tr>
<td>No. of Teeth</td>
<td>42</td>
<td>54</td>
</tr>
<tr>
<td>Transverse Module</td>
<td>3.00</td>
<td></td>
</tr>
<tr>
<td>Transverse Pressure Angle</td>
<td>20 deg</td>
<td></td>
</tr>
<tr>
<td>Transverse Tooth Thickness</td>
<td>4.00 mm</td>
<td>4.00mm</td>
</tr>
<tr>
<td>Face Width</td>
<td>15 mm</td>
<td></td>
</tr>
<tr>
<td>Hand</td>
<td>LEFT</td>
<td>RIGHT</td>
</tr>
<tr>
<td>Helix Angle</td>
<td>25 deg</td>
<td></td>
</tr>
<tr>
<td>Rack Tip Radius</td>
<td>0.001 mm</td>
<td>0.5 mm</td>
</tr>
<tr>
<td>Outside Diameter</td>
<td>132 mm</td>
<td>165 mm</td>
</tr>
<tr>
<td>Root Diameter</td>
<td>115 mm</td>
<td>150 mm</td>
</tr>
<tr>
<td>Inner Diameter</td>
<td>100 mm</td>
<td>140 mm</td>
</tr>
<tr>
<td>Young’s Modulus</td>
<td>2.06E5 N/mm²</td>
<td></td>
</tr>
<tr>
<td>Density</td>
<td>7.8e-9 Ns²/mm⁴</td>
<td></td>
</tr>
<tr>
<td>Template</td>
<td>FINEROOT.TPL</td>
<td></td>
</tr>
</tbody>
</table>

where,

\[
\begin{align*}
\text{Time} & - \text{Analysis step time in seconds} \\
\Theta_1 & - \text{Misalignment of pair 1} \\
ls_1 & - \text{Lead Slope of pair 1} \\
\Theta_2 & - \text{Misalignment of pair 2} \\
ls_2 & - \text{Lead Slope of pair 2} \\
\Theta_{31} & - \text{Misalignment of pair 3, group 1 (This occurs for SUN-PINION,RING-PINION,PINION-PINION pairs)} \\
ls_{31} & - \text{Lead Slope of pair 3, group 1} \\
\Theta_{32} & - \text{Misalignment of pair 3, group 2} \\
ls_{32} & - \text{Lead Slope of pair 3, group 2} \\
\end{align*}
\]

The output file for the analysis is shown below(Figure 10.4). \(-5.423673\text{E-3} \) radians is the estimated misalignment at the mesh. \(-5.921572\text{E-3} \) is the calculated lead slope per unit facewidth. The negative sign indicates the load shift to \(\zeta=-1\) end of the gear.
Figure 10.2: Misalignment definition - 3D view.
Figure 10.3: Misalignment definition - 2D view.

Figure 10.4: Misalignment File(HELICALMISALIGNMENT.DAT).
Figure 10.5: Contact Pressure Distribution of Base Model.
10.3 Calculation of Lead Slope

Using the above misalignment value, here are the steps to calculate lead slope correction for helical gears.

**Helical Gears**

**Step 1:** Calculate base helix angle of the gear

\[ \tan(\psi_b) = \tan(\psi) \times \cos(\phi_t) \]

where,

- \( \psi_b \) - Base Helix Angle
- \( \psi \) - Pitch Helix Angle
- \( \phi_t \) - Transverse Pressure Angle

\[ \psi_b = \arctan(\tan(25) \times \cos(20)) \]
\[ = 23.66235 \text{ deg} \]

**Step 2:** Calculate Lead Slope

\[ \text{LeadSlope} = \frac{\text{Misalignment}}{\cos(\psi_b)} \]
\[ = \frac{5.42367 \times 10^{-3}}{\cos(23.66235)} \]
\[ = 5.921515 \times 10^{-3} \]

**Step 3:** Amount of lead correction is

\[ \text{LeadCorrection} = \text{Facewidth} \times \text{LeadSlope} \]
\[ = 15.0 \times 5.921515 \times 10^{-3} \]
\[ = 0.0888227 \text{ mm} \]

For the spur gears, lead slope will be equal to the misalignment.

**Spur gears**

**Step 1:** Amount of lead correction is

\[ \text{Lead Correction} = \text{Facewidth} \times \text{Misalignment} \]

The lead correction is applied on one of the gears. In this example the lead correction is applied on gear 1 of rotor 1. The lead modification is applied using the LEADTABLE menu in TOOTH— > MODFN menu. Use the below inputs to apply a linear lead modification,

<table>
<thead>
<tr>
<th>NZETAS=2</th>
</tr>
</thead>
<tbody>
<tr>
<td>IZETA</td>
</tr>
<tr>
<td>1</td>
</tr>
<tr>
<td>2</td>
</tr>
</tbody>
</table>

The system is analyzed for one time with the above lead correction to gears. The contact pressure distribution is shown in Figure 10.6.
Figure 10.6: Contact Pressure Distribution of Corrected Model.
10.4 Results

The base model and the corrected model are analyzed for one full mesh cycle to show the contact pattern on the gears.

10.4.1 Contact Pattern

The contact patterns on gear 1 is shown in Table 10.2. The legend for the contour plot is in MPa.

Table 10.2: Contact Pattern on Gear 1

<table>
<thead>
<tr>
<th></th>
<th>Gear1</th>
</tr>
</thead>
<tbody>
<tr>
<td>Base Model</td>
<td>![Base Model Image]</td>
</tr>
<tr>
<td>Lead Corrected Model</td>
<td>![Lead Corrected Model Image]</td>
</tr>
</tbody>
</table>
ACOUSTIC ANALYSIS OF A 3D HOUSING USING TRANSMISSION3D AND COUSTYX

11.1 Transmission3D Model Setup

This example demonstrates the procedure for analyzing the dynamic response of a model in Transmission3D. We also show how to model the housing using a condensed housing matrix and use the dynamic response of the housing obtained in Transmission3D as an input to a Coustyx model for acoustic analysis of the housing.

Figure 11.1 shows the simple two-gear helical model setup in Transmission3D. The model consists of 3 rotors so we set NROTORS = 3 within the EDIT menu in Guide. We set NCONNECTORS = 15 to model the various bearings contained in the model. The bearing details are discussed later. NHOUSINGS = 1 enables the HOUSING menu, which allows us to model the condensed housing stiffness matrix. We use units consistent with the base units of Newtons (N), millimeters (mm), and seconds (s) throughout this example.

11.1.1 The Pinion and Gear Rotors

The sun and gear rotors are nearly identical and differ only in (1) the position of the rotor origin, (2) the hand of the helical gears, and (3) the presence of an extra shaft segment on the pinion shaft which connects a torsional stiffness bearing. The rotor menu inputs of the gear rotor are provided in Figure 11.2. The rotor origin of the pinion rotor is position at XPOSN = YPOSN = ZPOSN = 0. Both rotors are set to TYPE = OUTPUT, with TORQUE = 170000. We leave the reference frames of each rotor unconstrained in all degrees of freedom since each rotor shaft is supported by two bearings.
Figure 11.1: The single helical gear pair model.
Figure 11.2: The rotor menu.
11.1.1 The Pinion and Gear Rotor shafts

The shaft menu for the pinion and gear rotors is shown in Figure 11.3. Each of the two shafts begins at an AXIALPOSNSHAFT = -145.00 from the rotor origin. The material properties are those of steel: YOUNGSMOD = 207170, POISSON = 0.3, and DENSITY = 0.3. The damping is applied to the shafts using the Raleigh damping coefficients RALEIGHALPHA = 479 and RALEIGHBETA = 1.4E-07. Both the material properties and Raleigh damping values used here are consistent throughout the components in this example model. NSEGMENTS = 14 for the gear rotor, while the pinion rotor shaft contains NSEGMENTS = 15. Figure 11.4 and Figure 11.5 show the details of the gear and pinion rotor shafts, respectively.

11.1.2 The Pinion and Gear Teeth

The tooth details are entered within the SUN menu. Both gears’ mid-face is at the same axial position as the rotor origin, so AXIALPOSN = 0. We enter the tooth parameters within the TOOTH menu shown in Figure 11.6. The same parameters are entered for both the pinion and gear teeth with the exception of the HAND. The pinion HAND = RIGHT while the gear HAND = LEFT.

11.1.2 The Input Rotor

Since Transmission3D automatically constrains any rotor of the input TYPE, we cannot set the pinion or gear rotors to this type. Doing so would not allow us to obtain dynamic response results of the rotor specified as an INPUT. To work around this, we model a third rotor called the Input Rotor as an INPUT, since the model must have at least one INPUT and one OUTPUT. The input rotor connects to the pinion rotor through a torsional stiffness bearing in order to transfer torque. Selection of the torsional stiffness value is important and we discuss the selection process later.
Figure 11.4: The gear shaft details.
Figure 11.5: The pinion rotor shaft details.
Figure 11.6: The pinion and gear tooth menu.
Figure 11.7: The input rotor menu.

The input rotor origin is located at XPOS = YPOS = 0, ZPOS = 150.00 and the rotor axis points in the direction of the positive z-axis (AX = AY = 0, AZ = 1). We set the speed to RPM = 2100. The input rotor is constrained in all degrees of freedom by setting UXCONSTRAINT = UYCONSTRAINT = UZCONSTRAINT = THETAXCONSTRAINT = THETAYCONSTRAINT = TRUE.

The input rotor consists of just a single segment shaft, so ENABLESHAFTS = TRUE and NSHAFTS = 1. The shaft is modeled within the shaft menu, and the shaft details are provided in Figure 11.8.

11.1.3 Condensed Housing

The housing finite element model is modeled using a condensed mass and stiffness matrix in this example. Here, we present the process for condensing the FE model using the external FEA software package Hypermesh with Nastran’s Optistruct solver. The process using Abaqus CAE is detailed in the Dynamic Condensation of a Thin Plate chapter of this manual. If using another software package, consult the documentation for that package.

Follow the below steps for performing the dynamic condensation using Hypermesh w/Optistruct:

1. Build the FE mesh of the Housing [NOTE: Even if you plan to model the housing with shell elements you need to have solid elements at the bearing sleeves due to Transmission3D requirements.]

2. Use RBE2 rigid body elements (shown in white in Figure 11.9) to connect all dependent nodes on the inner diameter of the bearing sleeve to an independent node at the center of the sleeve. [Four such independent nodes are created for four bearing sleeves. These four nodes connect to the bearings in Transmission3D.]
3. Create a load collector with no card image for ASET. From Analysis > constraints. Select all related nodes and their dofs to constraint (while ASET load collector is current). ASET should contain nodes that form the interface connections (for example, four nodes created in (1) and their dofs connecting the bearings to the housing) as well as the nodes that would be constrained later on (SPCs).

4. Create a load collector with CMSMETH as the card image. Choose CBN (Craig-Bampton method) and set the Upper bound frequency, Number of modes and SPID. The SPID should be a unique node number not currently being used.

5. Now set the following control cards: (a) GLOBAL_CASE_CONTROL: Select CMSMETH and choose the load collector in (3), (b) PARAM, EXOUT, DMIGPCH (to write out condensed matrices: K2GG, M2GG, to punch file), (c) Unsupported PARAM - PARAM, EXCOUT, 3 (to write out OUT4 file with CMS modes).

6. The matrix condensation is performed using Optistruct. The Optistruct software is included within the Hypermesh package and is executed from Anaysis > Optistruct.

If the mesh does not pass the Optistruct mesh quality checks, the PARAM > CHECKELEM card can be used to bypass the suggested checks by setting the CHECKELEM_V1 parameter to NO. Alternatively, the ELEMQUAL card can be used to specify the range of acceptable values for the various quality checks and element types. The range for which a warning and/or error message is thrown can be adjusted for each element type and quality check. The information related to the quality checks performed on the model and the resulting values can be found in the .out file generated upon condensation of the model with Optistruct.

Execution of the Optistruct analysis yields multiple results files. For the purposes of this example, the following files are important:

1. HOUSING_AX.PCH: Punch file with condensed stiffness and mass matrices. This file is used in Transmission3D to model the Housing. NOTE: The “BOUNDARY GRID DATA” portion of this file must be deleted before running the dynamic T3D analysis.

2. HOUSING_X20A.OUT4: This contains the modes used to create the condensed matrices in the Component Mode Synthesis procedure. This file is output only when you define the control card - PARAM,EXCOUT,3 in the Optistruct input file. This data is used to back compute response at all nodes from the response at the interface nodes. Need this for the acoustic analysis in Coustyx.

3. HOUSING.DOFT: This contains the map between various node ids, dof and node type (regular dof or interface dof or modal dof). Need this for the acoustic analysis in Coustyx.
The condensed housing is modeled within Transmission3D’s HOUSING menu (Figure 11.10. The housing reference frame is fully constrained by checking UXCONSTRAINT, UYCONSTRAINT, UZCONSTRAINT, THETAXCONSTRAINT, THETAYCONSTRAINT, and THETAZCONSTRAINT. We set TYPE = NASTRAN_CONDENSED and enter FILENAME = HOUSING_AX.pch to match the output file generated from Optistruct. The STIFFMATNAME = KAAX and MASSMATNAME = MAAX must match the matrix names as they appear in the .pch file. The damping is applied within emphTransmission3D as Raleigh damping using the RALEIGHALPHA = 479 and RALEIGHBETA = 1.4E-07.

COORDXFM is the Calyx expression that is required to align the fixed housing reference frame with that of the Nastran model. In this case, we must translate the fixed reference frame by 75 mm along the y-axis, so we enter COORDXFM = Translate(75*e2). The number of nodes is equal to the number of nodes to which the ASET load collector was applied in (3). Thus, we set NNODES = 8.

The node data is modeled within the NODE submenu. The inputs for each node are provided in Table 11.1. Nodes 1-4 connect to the independent bearing nodes created in step 2 above. We set CONSTRAIN = TRUE for nodes 5-8 in order to tie the housing to the rotor reference frame.

### 11.1.4 Connectors

The CONNECTORS menu is shown in Figure 11.11. The model consists of NCONNECTORS = 15. Connectors 1-4 are the shaft bearings which constrain the pinion and gear rotors. Connectors 5-14 are reference bearings, which are zero stiffness bearings created only for data collection purposes at the connector origins. Connector 15 is the torsional stiffness bearing which connects the input rotor to the pinion rotor. Tables 11.2-11.4 provide the connectors menu input parameters for all 15 connectors.
Figure 11.10: The housing menu.

Table 11.1: Condensed housing NODE menu inputs.

<table>
<thead>
<tr>
<th>INODE</th>
<th>IDNODE</th>
<th>Constrained?</th>
<th>XNODE</th>
<th>YNODE</th>
<th>ZNODE</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>5742</td>
<td>NO</td>
<td>0.00</td>
<td>-75.00</td>
<td>-140.00</td>
</tr>
<tr>
<td>2</td>
<td>5743</td>
<td>NO</td>
<td>0.00</td>
<td>-75.00</td>
<td>140.00</td>
</tr>
<tr>
<td>3</td>
<td>5744</td>
<td>NO</td>
<td>0.00</td>
<td>75.00</td>
<td>-140.00</td>
</tr>
<tr>
<td>4</td>
<td>5745</td>
<td>NO</td>
<td>0.00</td>
<td>75.00</td>
<td>140.00</td>
</tr>
<tr>
<td>5</td>
<td>911</td>
<td>YES</td>
<td>-175.00</td>
<td>-225.00</td>
<td>130.00</td>
</tr>
<tr>
<td>6</td>
<td>946</td>
<td>YES</td>
<td>175.00</td>
<td>-225.00</td>
<td>130.00</td>
</tr>
<tr>
<td>7</td>
<td>2012</td>
<td>YES</td>
<td>175.00</td>
<td>-225.00</td>
<td>-130.00</td>
</tr>
<tr>
<td>8</td>
<td>2081</td>
<td>YES</td>
<td>-175.00</td>
<td>-225.00</td>
<td>-130.00</td>
</tr>
</tbody>
</table>
Figure 11.11: The connectors menu for the input rotor-pinion rotor torsional stiffness bearing.
Table 11.2: The connector origin and race positions.

<table>
<thead>
<tr>
<th>ID</th>
<th>XPOS</th>
<th>YPOS</th>
<th>ZPOS</th>
<th>AXPOS1 RACE1</th>
<th>AXPOS2 RACE1</th>
<th>DIA RACE1</th>
<th>AXPOS1 RACE2</th>
<th>AXPOS2 RACE2</th>
<th>DIA RACE2</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0.00</td>
<td>150.00</td>
<td>-140.00</td>
<td>-5.00</td>
<td>5.00</td>
<td>40.00</td>
<td>50.00</td>
<td>-5.00</td>
<td>5.00</td>
</tr>
<tr>
<td>2</td>
<td>0.00</td>
<td>150.00</td>
<td>140.00</td>
<td>-5.00</td>
<td>5.00</td>
<td>40.00</td>
<td>50.00</td>
<td>-5.00</td>
<td>5.00</td>
</tr>
<tr>
<td>3</td>
<td>0.00</td>
<td>0.00</td>
<td>-140.00</td>
<td>-5.00</td>
<td>5.00</td>
<td>40.00</td>
<td>50.00</td>
<td>-5.00</td>
<td>5.00</td>
</tr>
<tr>
<td>4</td>
<td>0.00</td>
<td>0.00</td>
<td>140.00</td>
<td>-5.00</td>
<td>5.00</td>
<td>60.00</td>
<td>60.00</td>
<td>-5.00</td>
<td>5.00</td>
</tr>
<tr>
<td>5</td>
<td>0.00</td>
<td>150.00</td>
<td>-102.12</td>
<td>-5.00</td>
<td>5.00</td>
<td>40.00</td>
<td>60.00</td>
<td>-5.00</td>
<td>5.00</td>
</tr>
<tr>
<td>6</td>
<td>0.00</td>
<td>150.00</td>
<td>-56.73</td>
<td>-12.50</td>
<td>12.50</td>
<td>40.00</td>
<td>60.00</td>
<td>-12.50</td>
<td>12.50</td>
</tr>
<tr>
<td>7</td>
<td>0.00</td>
<td>150.00</td>
<td>0.00</td>
<td>-10.00</td>
<td>10.00</td>
<td>2.00</td>
<td>0.50</td>
<td>-10.00</td>
<td>10.00</td>
</tr>
<tr>
<td>8</td>
<td>0.00</td>
<td>150.00</td>
<td>56.73</td>
<td>-12.50</td>
<td>12.50</td>
<td>40.00</td>
<td>60.00</td>
<td>-12.50</td>
<td>12.50</td>
</tr>
<tr>
<td>9</td>
<td>0.00</td>
<td>150.00</td>
<td>102.12</td>
<td>-5.00</td>
<td>5.00</td>
<td>40.00</td>
<td>60.00</td>
<td>-5.00</td>
<td>5.00</td>
</tr>
<tr>
<td>10</td>
<td>0.00</td>
<td>0.00</td>
<td>-102.12</td>
<td>-5.00</td>
<td>5.00</td>
<td>40.00</td>
<td>60.00</td>
<td>-5.00</td>
<td>5.00</td>
</tr>
<tr>
<td>11</td>
<td>0.00</td>
<td>0.00</td>
<td>-56.73</td>
<td>-12.50</td>
<td>12.50</td>
<td>40.00</td>
<td>60.00</td>
<td>-12.50</td>
<td>12.50</td>
</tr>
<tr>
<td>12</td>
<td>0.00</td>
<td>0.00</td>
<td>0.00</td>
<td>-10.00</td>
<td>10.00</td>
<td>2.00</td>
<td>0.50</td>
<td>-10.00</td>
<td>10.00</td>
</tr>
<tr>
<td>13</td>
<td>0.00</td>
<td>0.00</td>
<td>56.73</td>
<td>-12.50</td>
<td>12.50</td>
<td>40.00</td>
<td>60.00</td>
<td>-12.50</td>
<td>12.50</td>
</tr>
<tr>
<td>14</td>
<td>0.00</td>
<td>0.00</td>
<td>102.12</td>
<td>-5.00</td>
<td>5.00</td>
<td>40.00</td>
<td>60.00</td>
<td>-5.00</td>
<td>5.00</td>
</tr>
<tr>
<td>15</td>
<td>0.00</td>
<td>0.00</td>
<td>155.00</td>
<td>-5.00</td>
<td>5.00</td>
<td>40.00</td>
<td>60.00</td>
<td>-5.00</td>
<td>5.00</td>
</tr>
</tbody>
</table>

Table 11.3: Connector Stiffnesses

<table>
<thead>
<tr>
<th>ID</th>
<th>Name</th>
<th>$K_x$</th>
<th>$K_y$</th>
<th>$K_z$</th>
<th>$K_{\theta_x}$</th>
<th>$K_{\theta_y}$</th>
<th>$K_{\theta_z}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>GEARBRG1</td>
<td>5.00E+09</td>
<td>5.00E+09</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>2</td>
<td>GEARBRG2</td>
<td>5.00E+09</td>
<td>5.00E+09</td>
<td>1.00E+09</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>3</td>
<td>PINIONBRG1</td>
<td>5.00E+09</td>
<td>5.00E+09</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>4</td>
<td>PINIONBRG2</td>
<td>5.00E+09</td>
<td>5.00E+09</td>
<td>1.00E+09</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>15</td>
<td>INPUTBEARING</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>10</td>
</tr>
</tbody>
</table>

Table 11.4: Connector Damping

<table>
<thead>
<tr>
<th>ID</th>
<th>Name</th>
<th>$C_r$</th>
<th>$C_z$</th>
<th>$C_{\theta_x}$</th>
<th>$C_{\theta_z}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>GEARBRG1</td>
<td>50</td>
<td>0</td>
<td>0</td>
<td>700</td>
</tr>
<tr>
<td>2</td>
<td>GEARBRG2</td>
<td>50</td>
<td>50</td>
<td>0</td>
<td>700</td>
</tr>
<tr>
<td>3</td>
<td>GEARBRG1</td>
<td>50</td>
<td>0</td>
<td>0</td>
<td>700</td>
</tr>
<tr>
<td>4</td>
<td>GEARBRG2</td>
<td>50</td>
<td>50</td>
<td>0</td>
<td>700</td>
</tr>
<tr>
<td>15</td>
<td>INPUTBEARING</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>700</td>
</tr>
</tbody>
</table>
The PAIRS menu (Figure 11.12) is used to enter the contact grid parameters for the pinion-gear pair. The SEPTOL = 0.1 sets the tolerance for the tooth separation. Any two teeth separated by more than the SEPTOL are not considered to be in contact. NPROFDIVS sets the number of divisions on each side of the center of the tooth in the profile direction, while NFACEDIVS performs the analogous task in the tooth face direction. The remaining inputs are entered as shown in Figure 11.12.
### Table 11.5: Range menu parameters.

<table>
<thead>
<tr>
<th>RANGE</th>
<th>SOL METHOD</th>
<th>NTIME STEPS</th>
<th>DELTA TIME</th>
<th>START SPEED</th>
<th>START TORQUEFACTOR</th>
<th>END TORQUEFACTOR</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>STATIC</td>
<td>1</td>
<td>0.0000571429</td>
<td>0.00</td>
<td>1</td>
<td>1</td>
</tr>
<tr>
<td>2</td>
<td>NEWMARK</td>
<td>1</td>
<td>0.0000571429</td>
<td>0.00</td>
<td>1</td>
<td>1</td>
</tr>
<tr>
<td>3</td>
<td>NEWMARK</td>
<td>2000</td>
<td>0.0000571429</td>
<td>1.00</td>
<td>1</td>
<td>1</td>
</tr>
</tbody>
</table>

### 11.1.6 Setting Up and Running the Dynamic Analysis

The analysis parameters are entered within the SETUP menu. We would like to setup a dynamic study in which the system response reaches steady state. The analysis is split into 3 ranges, one static range and two dynamic ranges. The analysis parameters for each RANGE are entered within the range menu and are provided in Table 11.5.

We solve the static problem in the first range so that we can use the static solution as an initial condition for the dynamic study. This reduces the transient effects of the dynamic response since the starting value is closer to the steady state value. The single step dynamic range uses a ramp to slowly increase the speed to the desired final value, also reducing the transient effects and resulting in a response which reaches steady state more rapidly.

### 11.1.6.1 Analysis Results Files

Several relevant results files are generated by Transmission3D during the analysis. A brief description of each is provided below:

1. \(\text{\calyxtmp\dofmap.DAT}\): This file contains three sets of data, (i) $\text{DOFMAP}$ - list of interface dof node ids and their directions, (ii) $\text{SPCDOF}$ - list of constrained dofs, and (iii) $\text{RESFILE}$ - name of the file with the response data (e.g. HOUSINGDSP.DAT). This file is needed for the acoustic analysis in Coustyx. NOTE: The file HOUSINGDSP.DAT can be found in the working directory with other Helical3D outputs.

2. \(\text{\calyxtmp\extstruct.m}\): This is a matlab file that contains the mass (M) and stiffness (K) matrices of the condensed system. These values are actually read from HOUSING_AX.PCH file (generated from CMS analysis).

3. HOUSING1_DSP.DAT: This file contains the displacement response & forces at each dof of the condensed housing. NOTE: You will have response at the physical (interface dof) nodes as well as the modes used in CMS. Use dofmap.dat file to identify the corresponding dof for each column. The format for each row in HOUSING1_DSP.DAT looks like this:

<table>
<thead>
<tr>
<th>Time</th>
<th>Ux</th>
<th>Fx</th>
<th>Uy</th>
<th>Fy</th>
<th>Uz</th>
<th>Fz</th>
<th>Ux</th>
<th>Fx</th>
</tr>
</thead>
<tbody>
<tr>
<td>Index of the corresponding dof in dofmap.dat</td>
<td>(1)</td>
<td>(1)</td>
<td>(2)</td>
<td>(2)</td>
<td>(3)</td>
<td>(3)</td>
<td>(4)</td>
<td>(4)</td>
</tr>
</tbody>
</table>

4. GEARBRG_iRES.DAT, PINIONBRG_iRES.DAT (i=1,2,...7): These files contain responses at the gear and pinion bearings, respectively. The bearings at i = 1 and 2 refer to bearings that connect to the housing. The rest of the bearings are connected to ground (used to measure shaft deflections). Unlike the bearing at i = 1 (GEARBRG1.BRG), the bearings at i = 2 (GEARBRG2.BRG) has axial stiffness in addition to stiffness in x-y directions. The same is true for the bearings at i = 3 (PINIONBRG1.BRG) and i = 4 (PINIONBRG2.BRG) The format for each row in these files looks like this:

| Time | Ux | Uy | Uz | $\Theta_x$ | $\Theta_y$ | $\Theta_z$ | Fx | Fy | Fz | Mx | My | Mz |

5. PINION_ROTOR_RES.DAT and GEAR_ROTOR_RES.DAT: These files contain responses at the gear and pinion rotors. The file format is identical to the GEARBRG_iRES.DAT and PINIONBRG_iRES.DAT files.
11.1.6.2 Choosing the Torsional Stiffness Value  The selection of the torsional stiffness value is important since (1) too high of stiffness leads to undesired frequencies associated with the spring stiffness affecting the response and (2) too low of a value may introduce a very large magnitude of low frequency noise. The low frequency noise is unavoidable and cancels out when studying dynamic transmission error, but can be seen in the individual gear or pinion response as shown in Figure 11.13. Consideration needs to be taken to avoid these affects if running a long dynamic study. Depending upon the model setup, the addition of a torsional stiffness bearing may not be required. If three or more rotors make up the model, simply choose the rotor of interest to be either an OUTPUT or IDLER. The ‘torsional stiffness’ in a model of this nature would then depend upon the shaft length, stiffness, and constraint location.

11.2 Setting Up the Coustyx Model

The acoustic analysis is performed using Ansol’s Coustyx software package. Coustyx is a high performance acoustic analysis software that combines the Fast Multipole Method (FMM) technology with Boundary Element Methods (BEM) to solve steady state sound fields. Upon opening Coustyx we start a new model and select the ‘multidomain’ model type and ‘millimeter-Newton-seconds’ as the units system.

The steps for creating the Coustyx model are provided below. Refer to the Coustyx user’s manual at http://ansol.us/Manuals/CoustyxUsersManual.pdf for more details.

1. Import the mesh file. For this example, we import a Nastran bulk data file (.bdf). For Abaqus models, we would import the input file (.inp).

2. Fill the holes on the housing structure to prepare the structure for surface meshing.

3. Skin the housing structure to create the 2-D BE mesh.

4. Set the boundary boundary conditions. In this example, we define two boundary conditions: a rigid structure BC and a structure velocity BC. The boundary conditions are set by right-clicking on the ‘Boundary Conditions’
folder and selecting 'New'. The rigid BC is created by selecting the 'Uniform Normal Velocity' type and leaving the remaining options to their default values. The structure velocity is setup by choosing the 'Structure Velocity' boundary condition type and selecting 'HOUSING' from the structure name drop down box.

5. Select the side of the mesh on which the domain is by displaying the element orientation. In this case, the domain is on the POSITIVE side. We analyze the model in this example using a direct BE, unbounded, air domain. Chief points are automatically generated inside the housing by selecting 'Auto Generate' from the chief points dialogue box.

6. The analysis script is provided in the model directory and is titled AnalysisScript.txt. Right-clicking on 'Analysis Sequence' and selecting 'New' from the drop down box brings up the Analysis Sequence dialogue box. To set up the analysis, simply copy the script from the provided file and paste it within the text editor space under the 'Script' tab. The script contains 3 custom functions not available in the default analysis options. A description of each of these functions is given below:

(a) **LoadCondensedStructureXfmMatrixFromFile**(arg1, arg2, arg3, arg4): This function loads the transformation matrix (modes) used in CMS from output files generated by the FEA package used for condensation. Each of the four input arguments are strings. We describe each argument below:
   i. arg1: Name of the structure. The name should be the same as the mesh file name imported into Coustyx (w/o the extension)
   ii. arg2: File format type. Current supported types include "OptistructOUT4" type and "AbaqusMTX"
   iii. arg3: Name of the file that contains the modal matrix.
   iv. arg4 (optional): File with the map between the indices in modal matrix (loaded by arg3) to node ids and dofs

Usage:

```plaintext
var XfmMatRet = LoadCondensedStructureXfmMatrixFromFile('HOUSING', 'OptistructOUT4', 'HOUSING_X2OA.OUT4', 'HOUSING.DOFT');
```

Note: If loading an Abaqus model, arguments 3 and 4 will be the full filename with extension of the model recovery matrix, as this file includes both the modal matrix and the index mapping information.

(b) **LoadCondensedStructureGeneralizedCoordinatesFromFile**(arg1, arg2, arg3, arg4, arg5): This function loads the dynamic response of the Housing dofs from the Helical3D analysis. Or equivalently we could say that this function loads the generalized coordinates of the condensed housing. It also performs FFT to convert the time-domain results to frequency domain.

i. arg1: Name of the structure. The name should be the same as the mesh file name imported into Coustyx (w/o the extension)
ii. arg2: File type. For now only 'CalyxTimeDomain' type is supported
iii. arg3: Name of the file: dofmap.dat. NOTE: This file is located in the folder `\calyxtmp. Also note that you need to copy HOUSINGDSP.DAT file into to the same folder.
iv. arg4 (optional): Start time step for FFT if the results are in time domain
v. arg5 (optional): End time step for FFT if the results are in time domain

Usage:

```plaintext
var XModalRet = LoadCondensedStructureGeneralizedCoordinatesFromFile('HOUSING', 'CalyxTimeDomain', 'dofmap.dat', 600);
```

(c) **ComputeFrequencyResponseFromCondensedStructXfmMatrixAndGeneralizedCoords**(arg1, arg2): This function computes the frequency response from the transformation matrix (loaded from the CMS analysis results) and the generalized coordinates (loaded from the Helical3D dynamic analysis results).

i. arg1: Frequency of analysis
ii. arg2: Name of the structure

Usage:

```plaintext
var ForcedRespRet = ComputeFrequencyResponseFromCondensedStructXfmMatrixAndGeneralizedCoords(Eval(Freq), "HOUSING");
```
CHAPTER 12

BOLTED CONNECTION EXAMPLE

12.1 Transmission3D Model Setup

In many cases a bolted connection may be of interest so that the stresses in the bolt and/or the contact at the bolted connection can be examined. In this example, we demonstrate how to model a bolted connection between a gear rim and a diff cage, with the gear rim bolted to the flange of the carrier. An image of the model is shown in Figure 12.1. The model consists of two rotors, the Carrier Rotor and the Input Rotor. From the EDIT menu, we set NROTORS = 2. The model also includes contact surface pairs so we check ENABLEPAIRS and set NPAIRS to 3. The base units used in this example are Newtons (N), millimeters (mm), and seconds (s). All other units are derived from these three base units.

12.1.1 The Carrier Rotor

The carrier ROTOR menu is shown in Figure 12.2. The rotor is located in the global reference frame at XPOS = YPOS = ZPOS = 0, with a rotational axis pointing in the direction of the positive z-axis (AX = AY = 0, AZ = 1). We would like to specify the torque at this rotor, so we set the TYPE = OUTPUT and TORQUE = 3E+07. The rotor contains 7 shafts and 1 sun gear, so we set NSHAFTS = 7 and NSUNS = 1. The rotor is not supported by any bearings so we must constrain the rotor reference frame by setting UXCONSTRAINT, UYCONSTRAINT, UZCONSTRAINT, THETAXCONSTRAINT and THETAYCONSTRAINT to TRUE.

12.1.1.1 The Carrier

The carrier is modeled within the CARRIER menu. We set the TYPE = FECARRIER_NASTRAN which allows us to import the carrier as a series of Nastran bulk data files (.bdf). The model we would like to import contains seven separate BDF files so we set NCARRIERFILES to 7. The inputs for each file are entered into the FILE submenu. These inputs are provided in Table 12.1. In order to reduce the computer time required to build the carrier when running an analysis, we set USESECTORALSYMMETRY = TRUE. The BDF file we are importing contains a 45 degree sector of the entire carrier model, so we set NGROUPS = 8. The sector start surface begins at THETASTARTSECTOR = 0 degrees about the positive rotational axis using the right-hand rule.
Figure 12.1: The bolted connection model.

Figure 12.2: The carrier rotor menu.
Table 12.1: The FILE menu input parameters.

<table>
<thead>
<tr>
<th>FILE</th>
<th>NASTRAN FILE NAME</th>
<th>SUB TREE NAME</th>
<th>PREFERRED CUTTINGDIR</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>FrontJournalOuterSleeve.bdf</td>
<td>FrontJournalOuterSleeve</td>
<td>Z</td>
</tr>
<tr>
<td>2</td>
<td>FrontJournalInnerSleeve.bdf</td>
<td>FrontJournalInnerSleeve</td>
<td>Z</td>
</tr>
<tr>
<td>3</td>
<td>RearJournalOuterSleeve.bdf</td>
<td>RearJournalOuterSleeve</td>
<td>Z</td>
</tr>
<tr>
<td>4</td>
<td>RearJournalInnerSleeve.bdf</td>
<td>RearJournalInnerSleeve</td>
<td>Z</td>
</tr>
<tr>
<td>5</td>
<td>DiffCage.bdf</td>
<td>DiffCage</td>
<td>Z</td>
</tr>
<tr>
<td>6</td>
<td>GearRim.bdf</td>
<td>GearRim</td>
<td>Z</td>
</tr>
<tr>
<td>7</td>
<td>Bolt.bdf</td>
<td>CarrierBolt</td>
<td>Z</td>
</tr>
</tbody>
</table>

Figure 12.3: The carrier race menu.

In this example we do not model any pinions, so NPINIONS = 0. We set NRACES = 7 in order to create the 7 races that are symmetric about the carrier rotational axis. NINTERNALRACES is set to 6 in order to model the remaining connecting surfaces which are not symmetric about the carrier axis. NCONTACTSURFACES = 4 allows us to model the 4 contact surfaces on the journal bearing sleeves while, CONTACTPAIRS = 2 lets us define the two pairs between the 4 contact surfaces.

Carrier Races The carrier races are modeled within the RACE submenu of the carrier menu, shown in Figure 12.3. The carrier race details are provided in Figure 12.4. The two annular races are modeled using the CONICAL race type with a 90 degree cone angle. Each of the annular races connects to a single segment shaft. The outermost cylindrical race connects the carrier to the ring gear rim/shaft. The two smallest diameter cylindrical races each connect to a shaft and contact is modeled between the two in order to allow for a clearance between the bolt journal bearing sleeves. This shaft-shaft contact supports the gear in the radial direction. The last two races connect to shafts for constraint purposes.
Internal Races The internal race details are shown in Figure 12.5 and 12.6. Internal races are required when the races are symmetric about some other axis than the carrier rotational axis. In this example we need 6 internal races, two to connect each of the bolt flanges to the mating carrier surfaces, and 4 to connect the journal bearing sleeve surfaces to their mating bolt and carrier surfaces. Within the internal race menu, the race origin and orientation is first defined, then the race location is defined relative to this origin. Since the internal races connect two components within the carrier, both components should be checked upon generating the model to ensure nodes are found on both surfaces.

Contact Surfaces We model the contact between the two journal sleeves in order to transfer torque through the bolted connection. Doing so requires 4 contact surfaces, one at each of the journal sleeves. Each of the contact surfaces is cylindrical, with 2 of the surfaces being inside surfaces, and the other two being outside. Figures 12.5 and FigExample11InternalRacesContactSurfaces2 show the contact surface menu input details.

Contact Pairs The contact pairs are defined in the CONTACTPAIRS submenu of the CARRIER menu. The carrier contains NCONTACTPAIRS = 2, each of which models the contact between an inner and outer cylindrical surface on the two journal bearings. In the contact surface menu, each surface is given a SOLIDID. The contact pair is defined by selecting the two SOLIDID numbers of the two surfaces in contact as SURFACE1 and SURFACE2. NCONTACTDIVS sets the resolution level for the contact grid and must be between 1 and 4. In this case we set NCONTACTDIVS = 1.

12.1.1.2 The Carrier Rotor Shafts The carrier rotor consists of seven shafts (NSHAFTS = 7). The first shaft is the ring gear rim shaft shown in Figure 12.7. This shaft connects the ring gear to the carrier. The second and third shafts are placed between the two flanges on the carrier as shown in Figure 12.5. Each of the two shafts contains a contact surface so we use the FRONTCONTACT or BACKCONTACT feature as shown in Figure 12.8. The fourth and fifth shafts are the carrier support shafts. The details of these two shafts are provided in Figures 12.10 and 12.11, respectively. The locations of shafts six and seven are also shown in Figure 12.6. Shaft six connects to the ring gear flange on its
Figure 12.5: The carrier contact surface details.
Figure 12.6: The carrier contact surface details.
outer diametral surface, while the inner surface is defined as a contact surface by setting INSIDECONTACT = TRUE. Shaft 7 is set up in a similar manner, with the inside surface connected to the carrier and the outside surface set to OUTSIDECONTACT = TRUE. The shaft-shaft contact at the interface between shafts 6 and 7 keeps the ring gear flange radially aligned when the clearance at the interface is less than the bolt hole journal bearing clearance.

12.1.1.3 The Carrier Sun Gear  The carrier sun gear origin is coincident with the rotor origin (AXIALPOSN = 0). The gear connects to the outer diametral surface of the first carrier shaft. The tooth parameters are entered within the TOOTH menu shown in Figure 12.12.

12.1.2 The Input Rotor

The input rotor menu is shown in Figure 12.13. The input rotor consists of NSHAFTS = 1 and NSUNS = 1. The rotor origin is located within the global reference frame at XPOSN = 461.88, YPOSN = ZPOSN = 0.00. The rotor axis is oriented in the direction of the positive z-axis by setting AX = AY = 0, AZ = 1. The TYPE is set to INPUT and we choose RPM = 1000.

12.1.2.1 Input Rotor Shaft  The input rotor shaft is made up of a single segment as shown in Figure 12.14. The shaft contains a race on its outer diametral surface for connection with the inner surface of the input sun. We place a rigid shaft constraint on the shafts inner surface by selecting ODCONSTRAINT = TRUE and TYPE = RIGID.

12.1.2.2 Input Rotor Sun Gear  The input rotor sun gear input parameters are provided in the TOOTH menu shown in Figure 12.15.

12.1.3 Contact Pairs

This example consists of three contact pairs, two shaft-shaft and one gear contact pair. All three are defined within the EDIT > PAIRS menu. The first pair is between the front and back side of the two shafts with the axial holes. The PAIRS menu for this pair is shown in Figure 12.16. The second pair is the contact between input and carrier gear teeth and the PAIR MENU for this pair is shown in Figure 12.17. The third pair models the contact between the inner and outer surfaces of the two shafts between the carrier and sun gear flange.
Figure 12.8: The carrier shaft 2 segment menu.
Figure 12.9: The bolts hole pattern on carrier shafts 2 and 3.
Figure 12.10: The carrier shaft 4 details.
Figure 12.11: The carrier shaft 5 details.
<table>
<thead>
<tr>
<th>MultiX Edit Botor Sun Tooth</th>
<th>HELKANGLE</th>
<th>30.00000000000</th>
</tr>
</thead>
<tbody>
<tr>
<td>MODFN</td>
<td>RACKTFRAD</td>
<td>4.6582047000</td>
</tr>
<tr>
<td>NTEETH</td>
<td>PROTUSERANCE</td>
<td>0.0000000000</td>
</tr>
<tr>
<td>NFACCELEMS</td>
<td>OUTERDIA</td>
<td>765.0000000000</td>
</tr>
<tr>
<td>CODEORDER</td>
<td>ROUTDIA</td>
<td>719.0000000000</td>
</tr>
<tr>
<td>DISPLORDER</td>
<td>INNERDIA</td>
<td>684.0000000000</td>
</tr>
<tr>
<td>BEVELIZEDPROCESS</td>
<td>INNERCONEANGLE</td>
<td>0.00000000000+0.000</td>
</tr>
<tr>
<td>PROFFETYPE</td>
<td>DICHAMFER</td>
<td>0.00000000000</td>
</tr>
<tr>
<td>SYMMETRICTOOTH</td>
<td>ENABLEFRONTDIOFSET</td>
<td>0.00000000000</td>
</tr>
<tr>
<td>PLANE</td>
<td>ENABLERACKDIOFSET</td>
<td>0.00000000000</td>
</tr>
<tr>
<td>NORMALYDODULE</td>
<td>YOUNGSMOD</td>
<td>2060000.000000000</td>
</tr>
<tr>
<td>NORMALFACEANGLE</td>
<td>POISSON</td>
<td>0.00000000000</td>
</tr>
<tr>
<td>NORMALTHICKNESS</td>
<td>DENSITY</td>
<td>7.80000000000e-009</td>
</tr>
<tr>
<td>FACESWIDTH</td>
<td>ALPHA</td>
<td>479.00000000000</td>
</tr>
<tr>
<td>HAND</td>
<td>BETA</td>
<td>1.40000000000e-007</td>
</tr>
<tr>
<td></td>
<td>TEMPLATE</td>
<td>MEDIUM.TPL</td>
</tr>
</tbody>
</table>

Figure 12.12: The carrier sun gear tooth menu.
Figure 12.13: The input rotor menu.
Figure 12.14: The input rotor shaft details.
Figure 12.15: The input sun gear tooth menu.
Figure 12.16: The contact pairs menu for shaft-shaft contact.
Figure 12.17: The contact pairs menu for gear tooth contact.
CHAPTER 13

5 SPEED MANUAL TRANSMISSION EXAMPLE

This example describes the process used for modeling and analyzing a typical 5 speed manual transmission. The session file, 5SpeedManualTransmission.ses, contains the model data presented in this example and can be found in the SAMPLES/5SpeedManualTransmission directory on the Transmission3D technical support website: techsupport.ansol.com, along with the other necessary files for the model analysis. The model, shown in Figure 13.1, consists of 10 rotors (ENABLEROTORS = TRUE, NROTORS = 11), a housing (ENABLEHOUSINGS = TRUE, NHOUSINGS = 1), and 18 connectors (ENABLECONNECTORS = TRUE, NCONNECTORS = 18). The cut-away view of the model in Figure 13.2 shows all of the gears in the model with the exception of the reverse idler. There are two or three gear pairs in the model (NPAIRS = 2/3), depending upon the gear configuration being analyzed. We describe the setup of a baseline model in this example, and change the inputs required for running the model in each gear configuration using script files. The script files are also available in the the SAMPLES/5SpeedManualTransmission directory. The model inputs are entered within the EDIT menu shown in Figure 13.3. We use the metric engineering units system (N, mm, kg) throughout the example.
Figure 13.1: The 5 Speed Manual Transmission Model.
Figure 13.2: Cut-away view of the model components (reverse idler not pictured).
Figure 13.3: The EDIT menu.
13.1 The Input Rotor

The first rotor is the input rotor shown in Figure 13.4. The rotor is modeled within the ROTOR menu by setting ROTOR = 1, as shown in Figure 13.5. We decide to make the 1st rotor origin coincident with the global reference frame origin, so we set XPOSN = YPOSN = ZPOSN = 0. The rotor rotational axis is aligned with the positive z-axis of the global reference frame by setting AX = AY = 0, and AZ = 1. We set the rotor TYPE = INPUT since we would like to specify the speed into the transmission at this rotor (RPM = 500). The rotor is modeled with one shaft (ENABLESHAFTS = TRUE, NSHAFTS = 1) and one sun gear (ENABLESUNS = TRUE, NSUNS = 1). Two double tapered roller bearings support the rotor so we leave the rotor reference frame degrees of freedom unconstrained (UXCONSTRAINT = UYCONSTRAINT = UZCONSTRAINT = THETAXCONSTRAINT = THETAZCONSTRAINT = FALSE.)

13.1.1 The Shaft

The input rotor shaft inputs are entered into the SHAFT menu. The first segment of the shaft begins at an AXIAL-POSNSHAFT = -100 along the rotational axis of the rotor. The material properties used for the shaft are those of steel (YOUNGSMOD = 2.06E+05, POISSON = 0.3, and DENSITY = 7.8E-09). The same material properties are used for all components throughout the remainder of this example unless otherwise noted. The input shaft is modeled using NSEGMENTS = 4. The segments details are entered into the SEGMENT submenu.

The segment menu details for the input rotor shaft are provided in 13.6. The first segment contains a FLEXIBLE shaft constraint (ODCONSTRAINED = TRUE). The Fourier orders of interpolation of the displacement are CIRCORDER = 4 and AXIALORDER = 2. The shaft constraint ties the nodes at the outer surface of segment 1 to the rotor reference frame and the flexible constraint type allows for both rigid body and finite element deformation of the nodes at the surface. Alternatively, a RIGID constraint type would rigidly constrain the nodes to the reference frame and allow only rigid body motion of the surface nodes with respect to the rotor reference frame. Each rotor must contain one shaft constraint so that the shaft’s stiffness matrix is non-singular.

The second shaft segment has a race at its outer surface (ODRACE = TRUE) for connection to the input shaft bearing. Segment 3 contains a race at its outside surface to connect it to the input rotor sun gear, and the IDRACE at segment 4 connects the input rotor shaft to the front output shaft bearing.

Figure 13.4: The input rotor.
Figure 13.5: The ROTOR menu for the Input Rotor.
13.1.2 The Sun Gear

The input rotor sun gear (Figure 13.7) is modeled within the SUN menu. We select the axial position of the sun gear mid-face to be coincident with the rotor origin, so AXIALPOSN is set to 0. Within the BASE menu, the Fourier axial and circular orders of interpolation are set to AXIALORDER = 2 and CIRCORDER = 4, respectively. The tooth input parameters are entered into the TOOTH menu shown in Figure 13.8.
Figure 13.7: The input rotor sun gear.
Figure 13.8: The input rotor sun gear TOOTH menu.
13.2 The Countershaft Rotor

The countershaft rotor (Figure 13.9) is modeled within the ROTOR menu shown in Figure 13.10. The rotor origin is located at XPOSN = 0, YPOSN = -80, ZPOSN = 10 and its rotational axis points in the direction of the positive z-axis of the global reference frame (AX = AY = 0, AZ = 1). We set the TYPE to IDLER since neither a speed or a torque will be specified at this rotor. The rotor contains 5 shafts (ENABLESHAFTS = TRUE, NSHAFTS = 5), and 6 suns (ENABLESUNS = TRUE, NSUNS = 6). The rotor is constrained by two double tapered roller bearings so the rotor constraint boxes are all left unchecked.

13.2.1 The Countershaft Rotor Shafts

The countershaft rotor is modeled using 5 shafts by setting NSHAFTS = 5 within the ROTOR menu for rotor 2. Figure 13.11 shows the schematic for the shaft models. The SHAFT menu for the first shaft is shown in Figure 13.14. Shaft 1 consists of 9 segments (NSEGMENTS = 9), the first of which connects to the front countershaft bearing. Segments 2, 4, 6, and 8 have ODRACE surfaces enabled for connecting the input driven and 5th, 4th, and 3rd speed driver gears, respectively. The ENABLEFRONTINTERFACE option is turned on for this shaft as shown in Figure 13.14. This allows the front of the first shaft to connect to the back of the second shaft.

The second shaft is a single segment shaft (NSEGMENTS = 1) and contains an outer race for connection to the 2nd speed driver gear (ODRACE = TRUE), as well as front and back side races (ENABLEFRONTINTERFACE = ENABLEBACKINTERFACE = TRUE). The back side race connects the shaft to the front of the first shaft while the front side race connects to the back of shaft 3. Note, the circular order of the three races must be identical since nodes are shared between the 3 races at the outer surface of the shaft.

The third and fourth shafts are also modeled with a single segment (NSEGMENTS = 1). Shaft 3 contains an outer race for connection to a connector, as well as front and back side interfaces (ENABLEFRONTINTERFACE = ENABLEBACKINTERFACE = TRUE). The back interface connects the shaft to the front of both shaft 2 and sun gear number 3. The front interface connects to shaft 4. Shaft 4 contains an outer race (ODRACE = TRUE) for connection to sun gear number 2, as well as front and back races (ENABLEFRONTINTERFACE = ENABLEBACKINTERFACE = TRUE). The back and front interfaces allow us to connect the shaft to the front of shaft 3 and back of shaft 5, respectively.

Shaft 5 consists of two segments. The first segment contains an outer race to connect to a bearing, while the second segment contains a flexible constraint at its outer surface (ODCONSTRAINT = FLEXIBLE). ENABLEBACKINTERFACE = TRUE allows us to connect shaft 5 to shaft number 4. The remaining shaft inputs can be obtained from the schematic in Figure 13.13.
The countershaft rotor is a component of the rotor system in a turbomachinery context. The rotor is divided into multiple shafts, each with its own position and orientation.

**Figure 13.10: The countershaft Rotor menu.**

<table>
<thead>
<tr>
<th>SHAFT</th>
<th>SHAFT 1</th>
<th>SHAFT 2</th>
<th>SHAFT 3</th>
<th>SHAFT 4</th>
<th>SHAFT 5</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>AXIALPOSNSHAFT</strong></td>
<td>-50.0</td>
<td>152.5</td>
<td>167.5</td>
<td>190.0</td>
<td>230.0</td>
</tr>
<tr>
<td><strong>XYZ POSITION</strong></td>
<td>(0.0, 0.0, 0.0)</td>
<td>(0.0, 0.0, 0.0)</td>
<td>(0.0, 0.0, 0.0)</td>
<td>(0.0, 0.0, 0.0)</td>
<td>(0.0, 0.0, 0.0)</td>
</tr>
</tbody>
</table>

**Figure 13.11: The countershaft rotor shaft details.**

Rotor Origin = (0, -80, 10)

Axis Direction = (0, 0, 1)
Table 13.1: Countershaft sun gear axial positions.

<table>
<thead>
<tr>
<th>SUN GEAR</th>
<th>AXIALPOSN</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0</td>
</tr>
<tr>
<td>2</td>
<td>10</td>
</tr>
<tr>
<td>3</td>
<td>20</td>
</tr>
<tr>
<td>4</td>
<td>40</td>
</tr>
<tr>
<td>5</td>
<td>60</td>
</tr>
<tr>
<td>6</td>
<td>80</td>
</tr>
</tbody>
</table>

Figure 13.12: The countershaft rotor shaft 1 details.
Figure 13.13: The countershaft rotor shaft 2 through 5 details.
Figure 13.14: The countershaft rotor shaft 1 menu.
13.2.2 The Countershaf t Rotor Sun Gears

The SUN menu is used to model the 6 countershaft rotor sun gears. The first gear is the countershaft driven gear and it is located at AXIALPOSN = -10. The sun 1 details are shown in Figure 13.15. The BASE menu contains the inputs for the Fourier orders of interpolation at the base of the gear. These inputs must match the corresponding orders at the segment on the shaft to which the gear connects. The gear tooth input parameters are entered into the TOOTH menu shown in Figure 13.16.

The remaining 5 gears are modeled in a similar manner to the first gear. Figures 13.17 through 13.26 provide the gear details and TOOTH menu inputs for the remaining gears. Table 13.1 shows the axial positions of each gear’s mid-face.

The ENABLEFRONTDOFSET and ENABLEBACKDOFSET inputs are used for gears 2 and 3 in order to glue the front and back of the gears to the shaft interfaces described previously. The FRONTSHOULDERDIA and BACKSHOULDERDIA inputs set the maximum diameter of the nodes to connect to on the gear surfaces.
Table 13.16: The countershaft sun gear 1 TOOTH menu.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Helix Angle</td>
<td>00.000000000000</td>
</tr>
<tr>
<td>Rack Pitch</td>
<td>1.2473852080</td>
</tr>
<tr>
<td>Outer Dia</td>
<td>74.0000000000</td>
</tr>
<tr>
<td>Root Dia</td>
<td>64.2040900000</td>
</tr>
<tr>
<td>Inner Dia</td>
<td>53.2790000000</td>
</tr>
<tr>
<td>Inner Cone Angle</td>
<td>0.360000000000e+000</td>
</tr>
<tr>
<td>Diametrical Process</td>
<td></td>
</tr>
<tr>
<td>Profile Type</td>
<td>Simple</td>
</tr>
<tr>
<td>Symmetry Tooth</td>
<td></td>
</tr>
<tr>
<td>Plane</td>
<td>Transverse</td>
</tr>
<tr>
<td>Young Mod</td>
<td>2.0600000000</td>
</tr>
<tr>
<td>Poisson Ratio</td>
<td>0.3000000000</td>
</tr>
<tr>
<td>Density</td>
<td>7.6000000000e+008</td>
</tr>
<tr>
<td>Alpha</td>
<td>475.0000000000</td>
</tr>
<tr>
<td>Beta</td>
<td>1.4600000000e+007</td>
</tr>
<tr>
<td>Face Width</td>
<td>20.6000000000</td>
</tr>
<tr>
<td>Thermal Expansion</td>
<td>0.96000000000</td>
</tr>
<tr>
<td>Hand</td>
<td>Right</td>
</tr>
<tr>
<td>Template</td>
<td>MEDIUM_TFL</td>
</tr>
</tbody>
</table>

Figure 13.16: The countershaft sun gear 1 TOOTH menu.
Figure 13.17: The countershaft sun gear 2.
Figure 13.18: The countershaft sun gear 2 TOOTH menu.
Figure 13.19: The countershaft sun gear 3.
Figure 13.20: The countershaft sun gear 3 TOOTH menu.
Figure 13.21: The countershaft sun gear 4.
Figure 13.22: The countershaft sun gear 4 TOOTH menu.
Figure 13.23: The countershaf sun gear 5.
Figure 13.24: The countershaft sun gear 5 TOOTH menu.
Figure 13.25: The countershaft sun gear 6.
Figure 13.26: The countershaft sun gear 6 TOOTH menu.
13.3 The Driven Rotors

The transmission contains 6 driven gears, one for each speed configuration, that drive the output shaft when active. For the baseline model we model each gear as an INPUT, with RPM = 0. This keeps each driven rotor stationary and allows us to model one speed configuration at a time by changing the rotor TYPE to IDLER for the active gear. Each rotor sits on a bearing that is connected to the output shaft. A second connector, located between each driven rotor and the output shaft, acts as the clutch. The two connectors combined fully constrain each of the driven rotors, so the UXCONSTRAINT, UYCONSTRAINT, UZCONSTRAINT, THETAXCONSTRAINT, and THETAYCONSTRAINT inputs are all disabled for each rotor. Since the rotors are all modeled in a similar manner, we describe the setup of the 1st speed driven rotor in the following section and then present the inputs alone for the remaining driven rotors.
13.3.1 1st Speed Driven Rotor

The 1st speed driven rotor shown in Figure 13.27 is the third rotor. The ROTOR menu for the 1st speed driven rotor is shown in Figure 13.28. The rotor is positioned at XPOSN = YPOSN = 0, ZPOSN = 210, and its rotational axis is aligned with the positive z-axis of the global reference frame (AX = AY = 0, AZ = 1). The rotor is modeled with a shaft and a sun gear (NSHAFTS = 1, NSUNS = 1). Since we would like all of the driven rotors to initially be inactive in the baseline model, we set the rotor TYPE to INPUT and RPM = 0.

The 1st speed driven rotor shaft details are shown in Figure 13.29. The first segment of the shaft begins at AXIALPOSNSHAFT = -9.5, and the shaft is modeled using 3 shaft segments. The first segment has a race on the outer surface for connection to the 1st speed driven gear. Segment 2 contains the FLEXIBLE shaft constraint for the rotor, and segment 3 has a race at its inner diameter for connection to the torsional stiffness connector that acts as a clutch between the driven gear rotor and the output shaft.

The 1st speed driven sun gear shown in Figure 13.30 is modeled within the SUN menu. The AXIALPOSN of the mid-face of the gear is set to 0 so it is coincident with the rotor origin. The TOOTH menu shown in Figure 13.31 contains the tooth input parameters for the gear.
Figure 13.28: The 1st speed driven ROTOR menu.
Figure 13.29: The 1st speed driven rotor shaft details.
Figure 13.30: The 1st speed driven sun gear 1.

Figure 13.31: The 1st speed driven sun gear 1 TOOTH menu.
Figure 13.32: The 2nd speed driven rotor.

13.3.2 2nd Speed Driven Rotor
**Figure 13.33:** The 2nd speed driven ROTOR menu.
Figure 13.34: The 2nd speed driven rotor shaft details.
Figure 13.35: The 2nd speed driven sun gear 1.
Figure 13.36: The 2nd speed driven sun gear 1 TOOTH menu.
Figure 13.37: The 3rd speed driven rotor.

13.3.3 3rd Speed Driven Rotor
Figure 13.38: The 3rd speed driven Rotor menu.
Figure 13.39: The 3rd speed driven rotor shaft details.
Figure 13.40: The 3rd speed driven sun gear 1.
Figure 13.41: The 3rd speed driven sun gear 1 TOOTH menu.
13.3.4 4th Speed Driven Rotor

Figure 13.42: The 4th speed driven rotor.
Figure 13.43: The 4th speed driven ROTOR menu.
Figure 13.44: The 4th speed driven rotor shaft details.
Figure 13.45: The 4th speed driven sun gear 1.
Figure 13.46: The 4th speed driven sun gear 1 TOOTH menu.
13.3.5 5th Speed Driven Rotor

Figure 13.47: The 5th speed driven rotor.
### 5th Speed Manual Transmission Example

#### Figure 13.48: The 5th speed driven ROTOR menu.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>NAME</td>
<td>5THDRIVEN</td>
</tr>
<tr>
<td>XPOSN</td>
<td>0.00000000000×10^0</td>
</tr>
<tr>
<td>YPOSN</td>
<td>0.00000000000×10^0</td>
</tr>
<tr>
<td>ZPOSN</td>
<td>50.00000000000</td>
</tr>
<tr>
<td>AX</td>
<td>0.00000000000×10^0</td>
</tr>
<tr>
<td>AY</td>
<td>0.00000000000×10^0</td>
</tr>
<tr>
<td>AZ</td>
<td>1.00000000000</td>
</tr>
<tr>
<td>TYPE</td>
<td>INPUT</td>
</tr>
<tr>
<td>RPM</td>
<td>0.00000000000×10^0</td>
</tr>
<tr>
<td>ENABLESHAFTS</td>
<td>1</td>
</tr>
<tr>
<td>ENABLERUNS</td>
<td></td>
</tr>
<tr>
<td>ENABLERINGS</td>
<td></td>
</tr>
<tr>
<td>ENABLEBEVELS</td>
<td></td>
</tr>
<tr>
<td>ENABLEPODS</td>
<td></td>
</tr>
<tr>
<td>ENABLECARRIERS</td>
<td></td>
</tr>
<tr>
<td>DISPLAYTYPE</td>
<td>BODYFRAME</td>
</tr>
<tr>
<td>UXCONSTRAINT</td>
<td></td>
</tr>
<tr>
<td>FY</td>
<td>0.00000000000×10^0</td>
</tr>
<tr>
<td>UXCONSTRAINT</td>
<td></td>
</tr>
<tr>
<td>FZ</td>
<td>0.00000000000×10^0</td>
</tr>
<tr>
<td>UXCONSTRAINT</td>
<td></td>
</tr>
<tr>
<td>MX</td>
<td>0.00000000000×10^0</td>
</tr>
<tr>
<td>THETACONSTRAINT</td>
<td></td>
</tr>
<tr>
<td>MY</td>
<td>0.00000000000×10^0</td>
</tr>
<tr>
<td>THETA2</td>
<td>0.00000000000×10^0</td>
</tr>
<tr>
<td>REFERENCE_THETA2</td>
<td>0.00000000000×10^0</td>
</tr>
</tbody>
</table>
Figure 13.49: The 5th speed driven rotor shaft details.
Figure 13.50: The 5th speed driven sun gear 1.
Figure 13.51: The 5th speed driven sun gear 1 TOOTH menu.
13.3.6 The Reverse Rotors

The reverse gear configuration is modeled with the 1st speed/reverse countershaft gear driving the reverse idler gear. The reverse idler, in turn, drives the reverse driven gear to achieve the change in direction of rotation. The three mating reverse gears are shown in Figure 13.52. The reverse idler and driven gear are modeled as separate rotors since they rotate at different speeds. In the baseline model, both the reverse idler and reverse driven rotors are not engaged, so both are set to TYPE = INPUT with RPM = 0.
13.3.6.1 The Reverse Driven Rotor  

The ROTOR menu for the reverse driven rotor (rotor 8) is shown in Figure 13.53. The rotor origin is located at XPOSN = YPOSN = 0, ZPOSN = 230 in the global reference frame. One shaft and one sun gear are used to model the rotor, so we select ENABLESHAFTS = ENABLESUNS = TRUE and NSHAFTS = NSUNS = 1. The shaft details are shown in Figure 13.54 and the sun details are provided in Figures 13.55 and 13.56.
Figure 13.54: The reverse driven rotor shaft details.
Figure 13.55: The reverse driven sun gear 1.
Figure 13.56: The reverse driven sun gear 1 TOOTH menu.
13.3.6.2 The Reverse Idler Rotor  Rotor 9 is the reverse idler shown in Figure 13.57. The rotor origin is located at XPOS = 39.706, YPOSN = -75.156, ZPOS = 230. The rotor is modeled with one shaft and one sun gear so we set ENABLESHAFTS = ENABLESUNS = TRUE and NSHAFTS = NSUNS = 1. The rotor is fully constrained by a stiffness bearing so the rotor constraints are disabled in all degrees of freedom (UXCONSTRAINT = UYCONSTRAINT = UZCONSTRAINT = THETAXCONSTRAINT = THETAYCONSTRAINT = 0). The reverse idler shaft details are provided in Figure 13.58 and the idler gear inputs are shown in Figures 13.59 and 13.60.
Figure 13.58: The reverse idler rotor shaft details.

Figure 13.59: The reverse idler sun gear 1.
Figure 13.60: The reverse idler sun gear 1 TOOTH menu.
13.4 The Output Shaft Rotor

The output shaft rotor (Figure 13.61) is the tenth rotor. The rotor origin is located at XPOSN = YPOSN = 0, ZPOSN = 12.5. The rotor TYPE is set to OUTPUT with TORQUE = -2E+05. The rotor contains just a shaft (ENABLESHAFTS = TRUE, NSHAFTS = 1) and is unconstrained in all degrees of freedom as shown in the rotor menu displayed in Figure 13.62.

The output rotor shaft details are provided in Figure 13.63. The shaft rotor origin is coincident with the rotor origin, so AXIALPSONSHAFT is set to 0. The race at the outer surface of segment 1 connects to the front output shaft bearing, while race at segment 13 connects to the rear output shaft bearing. The races at the outer surfaces of segments 2, 4, 6, 8, 10, and 12 connect to the inner races of the support bearings for the 5th, 4th, 3rd, 2nd, 1st, and reverse driven rotors, respectively. The race at segment 3 connects to both the 4th and 5th speed clutch, while segments 5, 7, 9, and 11 connect to the 3rd, 2nd, 1st and reverse clutches, respectively. A flexible rotor constraint is placed on segment 14.
Figure 13.62: The output shaft rotor menu.
Figure 13.63: The output shaft details.
13.5 The Reverse Idler Pin Rotor

The reverse idler pin rotor, shown in Figure 13.64, is the eleventh rotor in the model. The rotor origin is located at $X_{\text{POSN}} = 39.706$, $Y_{\text{POSN}} = -75.156$, $Z_{\text{POSN}} = 222.5$ in the global reference frame as shown in the rotor menu (Figure 13.65). The rotor consists of one shaft ($NSHAFTS = 1$). The rotor type is set to ATTACHEDTOHOUSING, so that it deforms relative to the housing reference frame.

The reverse idler pin shaft, shown in Figure 13.66, begins at the rotor origin ($AXIAL_{\text{POSNSHAFT}} = 0$). The first and third shaft segments each contain a race at the outer diameter surface. The race at shaft 1 connects to the reverse idler support bearing, and the third segment race connects the pin shaft to the housing. Since the rotor is attached to the housing, a shaft constraint is not required. Both the housing and reverse idler pin are tied to the same reference frame by the constrained nodes on the housing.
Figure 13.65: The idler pin rotor menu.
Figure 13.66: The idler pin shaft details.
13.6 Housing

The finite element housing model, shown in Figure 13.67, is imported to Transmission3D as a Nastran bulk data file (.bdf). We use the Hypermesh FEA software package to apply SPC constraints at the nodes shown in Figure 13.68. We also apply the material properties of steel to the housing within the FEA software package, and then export the housing in .bdf file format.

The user input data for the Nastran housing is entered within the HOUSING menu (Figure 13.69) by selecting the NASTRAN_EXTERNALFE housing TYPE. We constrain the housing reference frame in all degrees of freedom by setting UXCONSTRAINT = UYCONSTRAINT = UZCONSTRAINT = THETA XCONSTRAINT = THETA YCONSTRAINT = THETA ZCONSTRAINT = TRUE. The filename of the Nastran BDF file is entered into the FILENAME field. The NODE TOLERANCE sets the maximum distance from a specified race location that will be used to find connecting nodes, and the MAXIMUMJOINTANGLE sets the maximum angle in degrees for smoothing two joining shell elements. We set these inputs to 0.001 and 15, respectively. The housing origin is coincident with the global origin of the model, so XSHIFT, YSHIFT and ZSHIFT are all 0.

The housing contains five races to connect to four of the shaft bearings and the idler support pin, as shown in Figure 13.70. The race inputs are entered into the RACE submenu of the housing menu. The RACE menu for the fourth housing race is shown in Figure 13.71.
Figure 13.68: The nodal housing constraints.

Figure 13.69: The housing menu.
Figure 13.70: The finite housing race locations.
Figure 13.71: The housing race menu.
13.7 Connectors

The transmission model consists of 18 connectors. Connectors 1 through 6 are cylindrical rolling element bearings that attach to the output shaft and each of the 6 driven rotors. Connectors 7 through 11 are double tapered roller bearings which provide support to the shafts. Connectors 7 and 8 connect the countershaft to the housing, connectors 9 and 10 connect the output shaft to the housing, and connector 11 provides support between the input and output shaft. The clutch connectors are connectors 12 through 17, and they allow for torque flow between the driven rotors and the output shaft when engaged. The clutches also assume the thrust load from the driven rotors to which they are connected. The reverse idler support bearing is the 18th connector. The idler bearing is modeled with a stiffness bearing to support the radial and thrust loads from the reverse idler gear. The connector inputs are entered into the CONNECTORS menu within the EDIT menu. Table 13.2 provides the race locations for each of the 18 connectors.

Table 13.3 shows the stiffness values for each of the stiffness bearings. Each clutch connector’s rotational stiffness about the rotational axis \( K(\theta_z) \) is set to zero for the base model setup. The analysis script file is used to adjust these values as needed based upon the desired speed configuration. The roller bearing GEOMETRY menu inputs are provided in 13.4.
### Table 13.2: Connector Race Locations

<table>
<thead>
<tr>
<th>ID</th>
<th>CONNECTOR</th>
<th>X</th>
<th>Y</th>
<th>Z</th>
<th>AX</th>
<th>AY</th>
<th>AZ</th>
<th>MEMBER1</th>
<th>AP1R1</th>
<th>AP2R1</th>
<th>DIA RACE1</th>
<th>MEMBER2</th>
<th>AP1R2</th>
<th>AP2R2</th>
<th>DIA RACE2</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>5THSPEED DRIVENBRG</td>
<td>0.0</td>
<td>0.0</td>
<td>50.0</td>
<td>0</td>
<td>0</td>
<td>1</td>
<td>ROTOR10</td>
<td>-7.5</td>
<td>7.5</td>
<td>30.0</td>
<td>ROTOR7</td>
<td>-7.5</td>
<td>7.5</td>
<td>45.0</td>
</tr>
<tr>
<td>2</td>
<td>4THSPEED DRIVENBRG</td>
<td>0.0</td>
<td>0.0</td>
<td>90.0</td>
<td>0</td>
<td>0</td>
<td>1</td>
<td>ROTOR10</td>
<td>-7.5</td>
<td>7.5</td>
<td>40.0</td>
<td>ROTOR6</td>
<td>-7.5</td>
<td>7.5</td>
<td>55.0</td>
</tr>
<tr>
<td>3</td>
<td>3RDSPEED DRIVENBRG</td>
<td>0.0</td>
<td>0.0</td>
<td>130.0</td>
<td>0</td>
<td>0</td>
<td>1</td>
<td>ROTOR10</td>
<td>-7.5</td>
<td>7.5</td>
<td>40.0</td>
<td>ROTOR5</td>
<td>-7.5</td>
<td>7.5</td>
<td>55.0</td>
</tr>
<tr>
<td>4</td>
<td>2NDSPEED DRIVENBRG</td>
<td>0.0</td>
<td>0.0</td>
<td>170.0</td>
<td>0</td>
<td>0</td>
<td>1</td>
<td>ROTOR10</td>
<td>-7.5</td>
<td>7.5</td>
<td>50.0</td>
<td>ROTOR4</td>
<td>-7.5</td>
<td>7.5</td>
<td>65.0</td>
</tr>
<tr>
<td>5</td>
<td>1STSPEED DRIVENBRG</td>
<td>0.0</td>
<td>0.0</td>
<td>210.0</td>
<td>0</td>
<td>0</td>
<td>1</td>
<td>ROTOR10</td>
<td>-7.5</td>
<td>7.5</td>
<td>50.0</td>
<td>ROTOR3</td>
<td>-7.5</td>
<td>7.5</td>
<td>65.0</td>
</tr>
<tr>
<td>6</td>
<td>REVERSE DRIVENBRG</td>
<td>0.0</td>
<td>0.0</td>
<td>230.0</td>
<td>0</td>
<td>0</td>
<td>1</td>
<td>ROTOR10</td>
<td>-7.5</td>
<td>7.5</td>
<td>50.0</td>
<td>Rotor8</td>
<td>-7.5</td>
<td>7.5</td>
<td>65.0</td>
</tr>
<tr>
<td>7</td>
<td>FRONT COUNTER SHAFTBRG</td>
<td>-80.0</td>
<td>-25.0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>1</td>
<td>Rotor2</td>
<td>-5.0</td>
<td>5.0</td>
<td>24.204</td>
<td>Housing1</td>
<td>-5.0</td>
<td>5.0</td>
<td>40.0</td>
</tr>
<tr>
<td>8</td>
<td>REAR COUNTER SHAFTBRG</td>
<td>-80.0</td>
<td>255.0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>1</td>
<td>Rotor2</td>
<td>-5.0</td>
<td>5.0</td>
<td>24.204</td>
<td>Housing1</td>
<td>-5.0</td>
<td>5.0</td>
<td>40.0</td>
</tr>
<tr>
<td>9</td>
<td>FRONT OUTPUT SHAFTBRG</td>
<td>-80.0</td>
<td>27.5</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>1</td>
<td>Rotor10</td>
<td>-5.0</td>
<td>5.0</td>
<td>30.0</td>
<td>Rotor1</td>
<td>-5.0</td>
<td>5.0</td>
<td>45.0</td>
</tr>
<tr>
<td>10</td>
<td>REAR OUTPUT SHAFTBRG</td>
<td>-80.0</td>
<td>252.5</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>1</td>
<td>Rotor10</td>
<td>-5.0</td>
<td>5.0</td>
<td>30.0</td>
<td>Housing1</td>
<td>-5.0</td>
<td>5.0</td>
<td>45.0</td>
</tr>
<tr>
<td>11</td>
<td>INPUT SHAFTBRG</td>
<td>-80.0</td>
<td>-25.0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>1</td>
<td>Rotor1</td>
<td>-5.0</td>
<td>5.0</td>
<td>30.0</td>
<td>Housing1</td>
<td>-5.0</td>
<td>5.0</td>
<td>45.0</td>
</tr>
<tr>
<td>12</td>
<td>5THSPEED CLUTCH</td>
<td>-80.0</td>
<td>59.0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>1</td>
<td>Rotor7</td>
<td>-0.5</td>
<td>0.5</td>
<td>50.0</td>
<td>Rotor10</td>
<td>-0.5</td>
<td>0.5</td>
<td>40.0</td>
</tr>
<tr>
<td>13</td>
<td>4THSPEED CLUTCH</td>
<td>-80.0</td>
<td>81.0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>1</td>
<td>Rotor6</td>
<td>-0.5</td>
<td>0.5</td>
<td>60.0</td>
<td>Rotor10</td>
<td>-0.5</td>
<td>0.5</td>
<td>40.0</td>
</tr>
<tr>
<td>14</td>
<td>3RDSPEED CLUTCH</td>
<td>-80.0</td>
<td>121.0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>1</td>
<td>Rotor5</td>
<td>-0.5</td>
<td>0.5</td>
<td>60.0</td>
<td>Rotor10</td>
<td>-0.5</td>
<td>0.5</td>
<td>40.0</td>
</tr>
<tr>
<td>15</td>
<td>2NDSPEED CLUTCH</td>
<td>-80.0</td>
<td>161.0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>1</td>
<td>Rotor4</td>
<td>-0.5</td>
<td>0.5</td>
<td>70.0</td>
<td>Rotor10</td>
<td>-0.5</td>
<td>0.5</td>
<td>50.0</td>
</tr>
<tr>
<td>16</td>
<td>1STSPEED CLUTCH</td>
<td>-80.0</td>
<td>201.0</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>1</td>
<td>Rotor3</td>
<td>-0.5</td>
<td>0.5</td>
<td>70.0</td>
<td>Rotor10</td>
<td>-0.5</td>
<td>0.5</td>
<td>50.0</td>
</tr>
<tr>
<td>17</td>
<td>REVERSE CLUTCH</td>
<td>-80.0</td>
<td>252.5</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>1</td>
<td>Rotor8</td>
<td>-0.5</td>
<td>0.5</td>
<td>30.0</td>
<td>Rotor10</td>
<td>-0.5</td>
<td>0.5</td>
<td>45.0</td>
</tr>
<tr>
<td>18</td>
<td>REVERSE IDLERBRG</td>
<td>39.706</td>
<td>-75.1557</td>
<td>230.0</td>
<td>0</td>
<td>0</td>
<td>1</td>
<td>Rotor9</td>
<td>-7.5</td>
<td>7.5</td>
<td>30.0</td>
<td>Rotor11</td>
<td>-7.5</td>
<td>7.5</td>
<td>16.0</td>
</tr>
</tbody>
</table>
Table 13.3: Connector Stiffnesses

<table>
<thead>
<tr>
<th>ID</th>
<th>CONNECTOR</th>
<th>$K_r$ (N/mm)</th>
<th>$K_z$ (N/mm)</th>
<th>$K_{\theta r}$ (N/mm/Rad)</th>
<th>$K_{\theta z}$ (N/mm/Rad)</th>
</tr>
</thead>
<tbody>
<tr>
<td>12</td>
<td>5THSPEED CLUTCH</td>
<td>0</td>
<td>1E+06</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>13</td>
<td>4THSPEED CLUTCH</td>
<td>0</td>
<td>1E+06</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>14</td>
<td>3RDSPEED CLUTCH</td>
<td>0</td>
<td>1E+06</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>15</td>
<td>2NDSPEED CLUTCH</td>
<td>0</td>
<td>1E+06</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>16</td>
<td>1STSPEED CLUTCH</td>
<td>0</td>
<td>1E+06</td>
<td>0</td>
<td>1E+06</td>
</tr>
<tr>
<td>17</td>
<td>REVERSE CLUTCH</td>
<td>0</td>
<td>1E+06</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>18</td>
<td>REVERSE IDLERBRG</td>
<td>1E+06</td>
<td>1E+06</td>
<td>1E+11</td>
<td>0</td>
</tr>
</tbody>
</table>

Table 13.4: Tapered Roller Bearing

<table>
<thead>
<tr>
<th>Connector</th>
<th>Name</th>
<th>No. of Rollers</th>
<th>Length</th>
<th>Roller Large Diameter</th>
<th>Cup Inner Diameter</th>
<th>Distance of Thrust Center</th>
<th>Cup Angle</th>
<th>Spread Thrust Center</th>
</tr>
</thead>
<tbody>
<tr>
<td>7</td>
<td>FRONTCOUNTERSHAFTBRG</td>
<td>15</td>
<td>7.5</td>
<td>5.0</td>
<td>37.5</td>
<td>0.0</td>
<td>10.0</td>
<td>5.0</td>
</tr>
<tr>
<td>8</td>
<td>REARCOUNTERSHAFTBRG</td>
<td>15</td>
<td>7.5</td>
<td>5.0</td>
<td>37.5</td>
<td>0.0</td>
<td>10.0</td>
<td>5.0</td>
</tr>
<tr>
<td>9</td>
<td>FRONTOUTPUTSHAFTBRG</td>
<td>15</td>
<td>10.0</td>
<td>5.0</td>
<td>45.0</td>
<td>0.0</td>
<td>15.0</td>
<td>5.0</td>
</tr>
<tr>
<td>10</td>
<td>REAROUTPUTSHAFTBRG</td>
<td>15</td>
<td>10.0</td>
<td>5.0</td>
<td>45.0</td>
<td>0.0</td>
<td>15.0</td>
<td>5.0</td>
</tr>
</tbody>
</table>
13.8 Pairs

The PAIRS menu is used to enter the gear contact pair input parameters. This example requires two pairs when any speed configuration other than the reverse configuration is analyzed. In the case where the reverse speed configuration is being analyzed, a third pair is required due to the presence of the reverse idler. The pairs menu in Figure 13.72 shows the input parameters for the input-countershaft driven gear pair. The 2nd and 3rd (if required) pairs menu inputs are entered by the script file corresponding to the desired speed configuration to be analyzed.
The analysis setup is accomplished with the use of a script file in this example. A separate script file for each speed configuration and contact pair is available in the SAMPLES/SSpeedManualTransmissionExample directory. The script file allows us to use the same baseline model session file described previously, alter inputs depending upon the speed configuration and gear pair of interest, and execute an analysis. The script file used for running an analysis in the first speed configuration for the output gear pair is shown in Section 13.9.1.

In the first speed configuration, we change the 2nd-5th and reverse driven gear bearings from ROLLER to STIFFNESS TYPE connectors. This is done for computational purposes as the absence of a load on the inactive rotors’ roller bearings results in the contact solver failing to converge to a solution. We also engage the clutch of the 1st speed driven rotor by setting the rotational stiffness value about the rotational axis ($K_{\theta z}$) to $1E+06$.

The countershaft driver-driven rotor gear pair changes depending upon the speed configuration, so we set this pair up within the script file. In the case of the first speed configuration, we pair rotor 3, sun 1 with rotor 2, sun 2. The rotor TYPE of the active driven rotor is also changed in the script file from the INPUT to IDLER. All of the inactive rotors remain INPUT rotors with RPM = 0. Setting the inactive rotors as inputs allows us to model the weight of the gears without having to solve for the contact at each of the inactive gears. This results in a significant reduction in computer time required to run an analysis.

The setup menu parameters are also included in the script file. The POSTPROCFILENAME and NTIMESTEPS input fields are set within this menu, as well as the DELTAT for the tooth cycle of interest. The script file presented here is for analyzing the output gear pair in the 1st speed configuration, so we determine the mesh cycle time and DELTAT value at the 1st speed countershaft-1st speed driven rotor gear pair. The following calculation was used to determine the DELTAT value in this case:

\[
\begin{align*}
RPM_{cs} &= RPM_{input} \times \left[\frac{N_{input}}{N_{csdriven}}\right] = 500 \times \left[\frac{36}{28}\right] = 642.857 RPM \\
T_{cycle} &= 60 / \left[\frac{RPM_{cs} \times N_{cs}}{T_{cycle}}\right] = 60 / \left[\frac{642.857 \times 12}{12}\right] = 0.00777777 \text{ sec} \\
DELTAT &= T_{cycle} / \left[\text{NTIMESTEPS} - 1\right] = 0.00777777 / \left[11 - 1\right] = 0.00077777 \text{ sec}
\end{align*}
\]

The script file then saves the modified session file under a new filename so the original baseline session file is preserved to use with another speed configuration script. The script file then generates the model, runs the analysis, and saves the results to a postprocessing folder with a name specific to the speed configuration. An iGlass results viewer file is also created. The sample script file for the 1st speed configuration is shown below.

The SAMPLES/SSpeedManualTransmission folder contains a total of 12 script files. There are two script files for each speed configuration: one that runs an analysis for the input gear pair and another that runs an analysis for the active output gear pair. The script files differ only in the DELTAT parameter that is used and the names of the session, postprocessing, and iGlass files that are generated. The generated files are saved with the ‘ip’ (input pair) or ‘op’ (output pair) specification, as well as the speed configuration for which the file represents.

### 13.9.1 First Speed Analysis Script

//Analysis Script for the First Gear Configuration - Output Gear Pair
SESFILE 5SpeedManualTransmission.ses LOADSES
EDIT

//Connectors: We change the inactive driven gear connectors to
//stiffness bearings so the contact solver does not fail on
//the lightly loaded roller bearings. This also reduces
//computer time required to analyze the model. The clutch
//connector of the active driven gear rotor is also engaged by
//entering a kthetaz stiffness value.

CONNECTOR

//SecondDrivenGearBrg
CONNECTOR 4
TYPE STIFFNESS
STANDARD TRUE
KR 1E+07
KTHETAR 1E+09

//ThirdDrivenGearBrg
CONNECTOR 3
TYPE STIFFNESS
STANDARD TRUE
KR 1E+07
KTHETAR 1E+09

//FourthDrivenGearBrg
CONNECTOR 2
TYPE STIFFNESS
STANDARD TRUE
KR 1E+07
KTHETAR 1E+09

//FifthDrivenGearBrg
CONNECTOR 1
TYPE STIFFNESS
STANDARD TRUE
KR 1E+07
KTHETAR 1E+09

//ReverseDrivenGearBrg
CONNECTOR 6
TYPE STIFFNESS
STANDARD TRUE
KR 1E+07
KTHETAR 1E+09

//FirstDrivenGearClutch
CONNECTOR 16
KTHETAZ 1E+06
KZ 1E+06
EXIT

//Pairs: The pairing of the active countershaft and driven output gears is done here. In the reverse speed configuration, 3 total pairs are defined. The 1st speed/reverse driver-reverse idler pair is pair 2 and the reverse idler-reverse driven output gear pair is pair 3.

NPAIRS 2
PAIRS
PAIR 2
SEPTOL 0.1
ADAPTIVEGRID ON
NPROFDIVS 3
NFACEDIVS 10
IROTOR1 3 ISUN1 1
IROTOR2 2 ISUN2 2
Rotor Type Setup: Here, the rotor type of the active driven gear rotor is changed from INPUT to IDLER. In the reverse gear configuration, we must change both the reverse idler and reverse driven gear types from INPUT to IDLER.

ROTOR
ROTOR 3
TYPE IDLER
EXIT
EXIT

Analysis Setup: The analysis setup parameters can be conveniently changed in the script file.

SETUP
POSTFILENAME postproc1ts_op_1stspeed.dat
CALYXVERBOSITY 5
RANGE
NTIMESTEPS 1
DELTATIME 0.0007777777
EXIT
EXIT

The session file for the specific speed configuration is saved so the baseline model file remains unchanged.

SESFILE 5SpeedManualTransmission_op_1stspeed.ses
SAVESES

The analysis is run and a postprocessing iGlass file is created.

GEN START POST POST postproc1ts_op_1stspeed.dat OK
GENIGLASS IGLASSFILE IGLASSPOST1ts_op_1stspeed.IGL START
EXIT EXIT QUIT OK
An automotive rear axle is a complex gear system, consisting of a hypoid gear set and a straight bevel differential set (Figure 14.1). We are going to show how to model such a system. We will also show how to include detailed models of a flexible housing and carrier in the analysis.

A schematic drawing of a rear axle system is shown in Figure 14.2. The power comes into the system through the propeller shaft. A hypoid pinion mounted on this shafts mates with a hypoid ring gear mounted on the carrier. The carrier has four straight bevel pinions that mate with the straight bevel gears on two half shafts. One of these half shafts drives the left wheel and the other drives the right wheel.

This example is described incrementally so that users can model the complete system step by step, or focus on a particular topic of interest (i.e. modeling a bevel planetary set). The model files are available in the SAMPLES/RearAxleTraining/directory on the Transmission3D technical support website techsupport.ansol.com.

We begin in the first section by modeling the hypoid pinion and ring gear set. In step 2, we introduce the carrier model and present the process for importing the carrier from an external FE mesh file. Step 3 introduces detailed roller bearing models to support the hypoid pinion and ring gear rotors. Step 4 presents the process for modeling the bevel planetary gear set, including the left and right wheel output shaft rotors with bevel gears. Step 4 also describes the addition of the planetary bevel pinion set with contact washers and pinion stiffness bearings, as well as stiffness connectors to support the half shafts. The 5th and final step documents the method used to import an external FE of the housing mesh. We use metric engineering units (N, mm, kg) throughout the example. All other units are derived from these base units.

The constraints in the model change through each step in the process since contact constraints are incrementally introduced. Therefore we describe the applicable rotor constraints and boundary conditions for each step separately. If conditions are not specified within a particular step, the user can assume the conditions to be identical to those of the previous step. This applies to rotor types, speed, torque, rotor constraints (ux, uy, uz, thetax, thetay), and rotor shaft constraints. The resulting models from each step are fully constrained, working models that can be run through pre- and post-processing.
Figure 14.1: A cut-away view of an automotive rear-axle assembly.
Figure 14.2: A schematic drawing of the automotive rear-axle assembly.
14.1 Step 1: Modeling the Hypoid Gear Pair

We begin by modeling a simple hypoid gear pair, shown in Figure 14.3. We do so by setting NROTORS = 2 and NPAIRS = 1 within the EDIT menu.

14.1.1 The Ring-Carrier Rotor

We name the rotor associated with the hypoid ring gear the “ring-carrier” rotor. To model this rotor, we set NSHAFTS and NHYPOIDS both to 1 within the rotor 1 menu (Figure 14.4). We do not model any bearings at this stage so the rotor needs to be fully constrained. We do this by selecting the UXC ONSTRAINT, UYCONSTRAINT, UZCONSTRAINT, THETAXCONSTRAINT, and THETAYCONSTRAINT check boxes. The rotor is therefore only allowed to rotate about its axis of symmetry. Since only two rotors are modeled in this step, we must define a torque on one and a speed on the other. For this rotor, we decide to set the TYPE = OUTPUT so that we can assign it a torque value. We choose a value of 1.0285714E+07 N/mm².
Figure 14.4: The ring carrier Rotor menu.
406  REAR AXLE TRAINING

14.1.1 The Hypoid Ring Gear

The hypoid ring gear is modeled within the HYPOID submenu of the ROTOR menu. The TYPE is set to CALYXMESH, which enables the user to import a hypoid set generated by the HYPOIDFACEMILLED / HYPOIDFACEHOBBED / HYPOIDK software or the Gleason-CageWin / Klingelnberg Kimos design software. AXIALPOSN is set to 0 so that the crossing point of the hypoid gears coincide with the rotor reference frame, and AXISDIRECTION is set to SAME to align the positive rotational axis of the gear reference frame with the rotor reference frame.

The gear_ip37.msh file contains the FE model information for one instance of the gear tooth mesh. The COMMON menu, shown in Figure 14.5 contains the remaining user inputs required to complete the hypoid gear. NTEETH sets the number of teeth, so that the program knows how many times to copy the tooth geometry specified in the .msh file. CIRCORDER and RADIALORDER specify the base surface Fourier series orders. RABASE, ZABASE, RBBASE, and ZBBASE are the radial (R) and axial (Z) positions of the front (B) and back (A) of the gear base surface. These values are generated along with the .msh file during it's generation and must be identical. The CONCAVE and CONVEX menus are used to model surface modifications of the gear tooth and any spacing errors. In this example we do not model either one, so we do not discuss these menus.
Figure 14.6: The ring carrier rotor hypoid gear details.
14.1.1.2 The Hypoid Gear Rim  

We use the RIM menu to model a webbed rim for the hypoid gear, as shown in Figure 14.7. The rim material properties are entered within the rim menu, as well as the number of elements in the circular direction, NTHETA. NSEGS sets the number of segments the user would like to use to model the rim. In this example we set NSEGS = 4.

The SEGRIM menu contains the inputs related to each of the 4 rim segments. POSN defines the segment position as either INSIDE, OUTSIDE, BEHIND, or AHEAD. Figure 14.9 shows the definitions for each of these location types. NDIVSETA and NDIVSZETA sets the number of element divisions in the eta and zeta directions. The eta and zeta directions are defined by the segment reference frames show for each POSN in Figure 14.9. RA, ZA, RB, and ZB set the radial and axial distances of the toe and heel side nodes of the rim as shown in Figure 14.7 (A-toe, B-heel). The R and Z values used to create the hypoid gear rim in this example are provided in Figure 14.7. Race surfaces are automatically generated on the outside surface of the first rim segment for interface with the base surface of the gear and the inside surface of the last rim segment for interface with the rotor shaft. These two races use the Fourier orders specified in the HYPOID and shaft SEGMENT menus, respectively.

Figure 14.7: The ring carrier rotor hypoid gear rim details.
Figure 14.8: The ring carrier rotor hypoid gear rim menu.
Figure 14.9: The hypoid rim segment position definitions.
### 14.1.1.3 The Ring Carrier Rotor Shaft

The ring carrier rotor shaft is included to provide us with a location to define a shaft constraint. The shaft constraint ties the nodes at the selected shaft surface to the rotor reference frame and fully constrains the rotor FE mesh. We model the shaft within the SHAFT menu shown in Figure 14.10. The shaft begins at an AXIALPOSNSHAFT = 91.416 from the rotor origin, and it made up of 1 shaft segment (NSEGMENTS = 1). We used the default steel material properties throughout this model (YOUNGSMOD = 206000, POISSON = 0.300, & DENSITY = 7.8E-09). Setting ENABLEBACKINTERFACE = TRUE creates a race surface on the back side (negative rotational axis direction) of the shaft for connection to the ring gear rim. BACKCIRCORDER = 4 and BACK RADIALORDER = 2 sets the circular and radial Fourier orders of interpolation for the race surface.

The shaft segment details are entered within the SEGMENT menu. Figure 14.11 provides many of the SEGMENT menu inputs graphically. We define the shaft constraint at the inner surface by selecting IDCONSTRAINED = TRUE, IDTYPE = RIGID. The cylindrical race where the back of the shaft connects to the front of the hypoid rim is also shown in Figure 14.11.
Figure 14.11: The ring carrier rotor SHAFT details.
14.1.2 The Propshaft Rotor

The second rotor in this iteration is the named the "propshaft rotor". Within the ROTOR menu (Figure 14.12), we set the rotor reference frame origin to XPOSN = ZPOSN = 0, YPOSN = -43.983. These coordinates place the rotor origin at the hypoid crossing point minus the pinion offset (offset = -43.983). The rotational axis of the rotor is aligned with the negative x-axis of the global reference frame (AX = -1, AY = AZ = 0). We set the rotor TYPE = INPUT for this rotor, since we want to specify the speed. The speed is set to RPM = -1000. The rotor consists of one shaft (NSHAFTS = 1) and 1 hypoid (NHYPOIDS = 1). Similarly to the ring-carrier rotor, we constrain the rotor reference frame rigid body motion in all degrees of freedom except about its axis of symmetry (UXCONSTRAINT = UYCONSTRAINT = UZCONSTRAINT = THETAXCONSTRAINT = THETAYCONSTRAINT = THETAZCONSTRAINT = TRUE).
14.1.2.1 The Propshaft Rotor Shaft

The shaft is modeled within the SHAFT menu shown in Figure 14.13. The shaft is positioned at an AXIALPOSNSHAFT = 95.00 from the rotor origin and consists of 11 segment (NSEGMENTS = 11). The inputs of each segment are entered within the SEGMENT menu. The shaft segment details are shown in the schematic in Figure 14.14.
Figure 14.14: The propshaft rotor shaft details.
14.1.2.2 The Hypoid Pinion and Rim  The hypoid pinion is modeled within the HYPOID menu in a similar manner to the hypoid gear described previously. The COMMON menu and associated inputs are shown in Figure 14.15. Figure 14.17 shows the radial and axial positions of the base surface of the pinion gear.

The hypoid pinion rim is modeled with a single segment. The rim model inputs of Figure 14.17 are entered within the RIM and SEGMENT menus.
STEP 1: MODELING THE HYPOID GEAR PAIR

**Figure 14.16:** The hypoid pinion details.

- **Base surface of gear tooth mesh**
- **RABASE**: 15.7731
- **ZABASE**: 134.2064
- **RBBASE**: 30.3801
- **ZBBASE**: 202.9269

**Origin of Rotor Axis**
- Pinion Crossing Point
- {0, -43.983, 0}

**AXIALPOSN**: 0.0

**Rotor Axis Orientation**
- (-1, 0, 0)

**Hypoid Axis Orientation**
- SAME as rotor

**Tooth Finite Element Model**
- MESHFILE: “pinion_LP37.msh”
- NTEETH: 7
Figure 14.17: The hypoid pinion rim details.
In step one of the model, we need only one pair between the hypoid pinion and gear, so we set NPAIRS = 1 within the EDIT menu. Figure 14.18 shows the PAIRS menu and associated inputs. We set the number of divisions over the semi-width in the profile direction as NPROFDIVS=1 (total # of divisions = 2 * NPROFDIVS + 1). The number of divisions in the face direction is set as NFACEDIVS = 3, which is the total number of divisions across the face of the gear. The width of the contact grid is determined by the DSPROF input. For this case, we leave the value at the default value of 0.2. More detailed information on selecting this value can be found in the Pairs section of Wind Turbine chapter.
14.2 Step 2: Importing the FE Carrier

In Step 2, we import the FE carrier into the model and connect it to the hypoid ring gear modeled in the previous step. The carrier in this step does not contain any pinions so it acts as a shaft connected to the rotor. We connect the hypoid gear rim directly to the carrier so the hypoid gear shaft described in step 1 is no longer needed. The rotor will still contain a shaft for the purpose of constraining the rotor to its reference frame, but the shaft will now be attached to the FE carrier model. Figure 14.19 shows the updated ROTOR menu for rotor 1. The ENABLESHAFTS box is checked with NSHAFTS still set to 1, and the ENABLECARRIERS box is now checked and NCARRIERS set to 1.

14.2.1 The Carrier Menu

The carrier menu, shown in Figure 14.20, contains all of the input fields for the FE carrier model. The TYPE = FECARRIER_NASTRAN field specifies that the carrier will be imported as an FE mesh using a Nastran bulk data file (.bdf) format. In this example, only one file is imported, so NFECARRIERFILES is set to 1. NODETOLERANCE sets the maximum distance from a defined race or contact surface location that will be included as a node on the specified surface. MAXJOINTANGLE sets the maximum angle for edge smoothing and is 15 degrees by default. The carrier origin is shifted along the rotor axis by AXIALSHIFT = -89.016. NRACES sets the number of races to define on the carrier for connection to another shaft or gear base surface on the rotor. We set NRACES = 2 in order to join the carrier to the hypoid gear rim race surface, and the carrier to a the rotor shaft. NINTERNALRACES sets the number of internal races, or races which join two components within the same imported FE carrier file. There are four locations where the spider pin joins with the carrier, so NINTERNALRACES is set to 4.

The carrier file information is entered into the FILE submenu. Within the file menu carrier_constraints.bdf is entered as the NASTRANFILENAME. The PREFERREDCASTINGDIR is set to Z, since that is the carrier rotational axis from the FE mesh file reference frame. The BDF file is located within the SAMPLES/RearAxleTraining/directory on the Transmission3D tech support website.
The INTERNALRACE and RACE menus are used to enter the race information. Figure 14.21 shows the location of the standard races modeled within the RACE menu. Race 1 is of TYPE=CONICAL and connects the hypoid gear base surface to the carrier. Race 2 is a CYLINDRICAL race and connects the carrier to the rotor shaft. The carrier BDF file contains the meshes for both the carrier and the spider pin. Internal races connect the spider pin to the holes on the carrier at the four locations specified in Table 14.1.
Figure 14.21: The carrier RACE menu details for Step 2.

Table 14.1: FE Carrier Internal Race Menu Input Data

<table>
<thead>
<tr>
<th>Race</th>
<th>XPOS</th>
<th>YPOS</th>
<th>ZPOS</th>
<th>DIA</th>
<th>AXPOSN1</th>
<th>AXPOSN2</th>
<th>CIRC ORDER</th>
<th>AXIAL ORDER</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>82.00</td>
<td>0.00</td>
<td>-24.72</td>
<td>25.934</td>
<td>-12.00</td>
<td>12.00</td>
<td>1</td>
<td>1</td>
</tr>
<tr>
<td>2</td>
<td>0.00</td>
<td>82.00</td>
<td>-24.72</td>
<td>25.934</td>
<td>-12.00</td>
<td>12.00</td>
<td>1</td>
<td>1</td>
</tr>
<tr>
<td>3</td>
<td>-82.00</td>
<td>0.00</td>
<td>-24.72</td>
<td>25.934</td>
<td>-12.00</td>
<td>12.00</td>
<td>1</td>
<td>1</td>
</tr>
<tr>
<td>4</td>
<td>0.00</td>
<td>-82.00</td>
<td>-24.72</td>
<td>25.934</td>
<td>-12.00</td>
<td>12.00</td>
<td>1</td>
<td>1</td>
</tr>
</tbody>
</table>
STEP 3: ADDING ROLLER BEARINGS TO THE HYPOID PINION AND GEAR ROTORS

14.2.2 The Ring-Carrier Rotor Shaft

The rotor shaft is modeled within the SHAFT submenu of the ROTOR menu. The starting point of the first shaft segment is positioned by entering AXIALPOSNSHAFT=-32.92. The shaft consists of just a single segment, so NSEGMENTS is set to 1. The SEGMENT menu, shown in Figure 14.22, is used to enter the details of the shaft segment. The inner surface of the cylindrical segment connects to the carrier so we set IDRACE=TRUE, and match the mating carrier race orders of interpolation by setting CRICORDERINNER=4 and AXIALORDER=2. On the outer surface, we constrain the nodes to the rotor race by setting ODCONSTRAINED=TRUE and ODTYPE=FLEXIBLE. This satisfies the rotor constraint requirement for the carrier-hypoid gear rotor.

14.3 Step 3: Adding Roller Bearings to the Hypoid Pinion and Gear Rotors

Roller bearing connectors are added in this step to support the two rotors modeled in steps 1 and 2. Selecting the ENABLECONNECTORS box and setting NCONNECTORS = 5 in the EDIT menu activates the five connector instances within the CONNECTOR menu. We must also define the race surfaces to which the connectors will attach. We can also remove all rotor constraints within the rotor 1 and rotor 2 ROTOR menus since the 5 bearings sufficiently constrain both rotors in all degrees of freedom.
14.3.1 The Ring Carrier Rotor Connector Races

The carrier support bearings connect the carrier rotor to ground, so we need to add 2 new races on the carrier for the bearings. We do this by changing NRACES from 2 to 4 within the carrier RACE menu and entering the additional race details shown in Figure 14.23.
STEP 3: ADDING ROLLER BEARINGS TO THE HYPOID PINION AND GEAR ROTORS

Figure 14.23: The hypoid ring carrier race locations, step 3.
14.3.2 The Propshaft Rotor Connector Races

The location of the bearing races on the prop shaft rotor shaft are shown in the updated shaft schematic in Figure 14.24. The races are enabled by simply checking the ODRACE box for segments 1, 4, and 6 and selecting the Fourier series orders for modeling the deformation at the race interface. In this example, we use CIRCORDER = 8 and AXIALORDER = 2 for all three shaft segments.
### Table 14.2: Connector Locations and Orientations

<table>
<thead>
<tr>
<th>Connector</th>
<th>Name</th>
<th>X</th>
<th>Y</th>
<th>Z</th>
<th>A_x</th>
<th>A_y</th>
<th>A_z</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>CARRIERBRG1</td>
<td>0</td>
<td>0</td>
<td>-38.66354</td>
<td>0</td>
<td>0</td>
<td>1</td>
</tr>
<tr>
<td>2</td>
<td>CARRIERBRG2</td>
<td>0</td>
<td>0</td>
<td>154.67064</td>
<td>0</td>
<td>0</td>
<td>1</td>
</tr>
<tr>
<td>3</td>
<td>PINIONCYLBRG</td>
<td>-105</td>
<td>-43.983</td>
<td>0</td>
<td>-1</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>4</td>
<td>PINIONTAILBRG</td>
<td>-287.9858</td>
<td>-43.983</td>
<td>0</td>
<td>-1</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>5</td>
<td>PINIONHEADBRG</td>
<td>-200.5326</td>
<td>-43.983</td>
<td>0</td>
<td>-1</td>
<td>0</td>
<td>0</td>
</tr>
</tbody>
</table>

Figure 14.25: The GEOMETRY menu details for connector 3.

### 14.3.3 The Connectors Inputs

The roller bearing inputs are entered within the CONNECTORS menu. Table 14.2 shows the reference frame origins and axis orientations for each of the roller bearing connectors. Connectors 1 and 2 connect the carrier to ground, while connectors 3 through 5 connect the propellor shaft to ground.

Figure 14.25 shows the connector menu for the cylindrical roller bearing connector 3. The remaining 4 connectors are all of the TAPERED roller bearing type. The tapered roller bearing GEOMETRY menu inputs are shown in Figure 14.26 and Table 14.3.

The remaining submenus of the CONNECTORS menu can be filled in using the default values for the purposes of this example.
Table 14.3: Tapered Roller Bearing

<table>
<thead>
<tr>
<th>Connector</th>
<th>Name</th>
<th>No. of Rollers</th>
<th>Length</th>
<th>Roller Large Diameter</th>
<th>Cup Inner Diameter</th>
<th>Dist Thrust Center</th>
<th>Axial Clearance</th>
<th>Cup Angle</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>CARRIERBRG1</td>
<td>30</td>
<td>25</td>
<td>12</td>
<td>111.0992</td>
<td>35.7045</td>
<td>-0.01</td>
<td>-21.9828</td>
</tr>
<tr>
<td>2</td>
<td>CARRIERBRG2</td>
<td>30</td>
<td>25</td>
<td>12</td>
<td>111.0992</td>
<td>35.7045</td>
<td>-0.01</td>
<td>21.9828</td>
</tr>
<tr>
<td>4</td>
<td>PINIONTAILBRG</td>
<td>18</td>
<td>14</td>
<td>15</td>
<td>91.1854</td>
<td>26.8503</td>
<td>-0.01</td>
<td>-23.3291</td>
</tr>
<tr>
<td>5</td>
<td>PINIONHEADBRG</td>
<td>18</td>
<td>35</td>
<td>20</td>
<td>120.8478</td>
<td>44.1997</td>
<td>-0.01</td>
<td>23.0243</td>
</tr>
</tbody>
</table>

Figure 14.26: The GEOMETRY menu for the TAPERED roller bearing type.
In Step 4, we add the bevel planetary gear set to the ring-carrier rotor and also add the two half shaft rotors, as shown in Figure 14.27. The planetary set consists of 4 bevel pinions, washers and stiffness bearings between the pinions and pin shafts. Each of the half shafts is modeled with a bevel gear and three shafts. Two connectors support each half shaft, one between the carrier and the shaft and the other between the shaft and ground. The half shaft connectors are modeled as a simplified stiffness bearings. At this stage, we must also change the rotor TYPE for the carrier rotor to IDLER, since we would rather specify the reaction torque at one of the wheels. The propshaft rotor remains an INPUT rotor.
Figure 14.28: The CARRIER menu.

14.4.1 The Ring-Carrier Rotor with Bevel Pinions

The bevel planetary pinions are added to the model by selecting ENABLEPINIONS within the rotor 1 CARRIER menu shown in Figure 14.28. Upon selecting the ENABLEPINIONS option, the NPINIONS and NGROUPS input fields appear. We set NPINIONS = 1 and NGROUPS = 4 in the pinion menu. This merely means that there is only one kind of pinion on the carrier, even though there are 4 identical copies (NGROUPS=4) of this pinion. The result is 4 copies of the pinion, each spaced spaced by $\frac{360}{4} = 90$ degrees.
14.4.2 The Bevel PINION

The pinion THETAPOSITION is set within the PINION menu and defines the starting position of the first pinion group following the right hand rule about the carrier axis. AXPOSN sets the position of the pinion crossing point on the carrier rotational axis. PHIPOSN defines the angle at which the bevel pinion axis intersects the carrier rotational axis, so we set this to 90 for this example. The Fourier series orders of interpolation for the base surface of the bevel pinions, CIRCORDER and AXIALORDER, are set to 4 and 2, respectively. The same orders should be used for the pin race surfaces defined within the PINRACE menu. Figure 14.29 shows the PINION menu with the remaining inputs used in this example.

**14.4.2.1 The Bevel Pinion RACE**  Figure 14.30 shows the inputs of the PINRACE menu. This menu defines the connecting surfaces between the cross member and carrier. Just the one race needs to be defined since there is only one pinion. The races for the four groups are generated similarly to the pinion copies by specifying PHIPOSN and NGROUPS.

**14.4.2.2 The Bevel Pinion WASHER**  A washer is modeled in WASHER sub-menu when ENABLEWASHER is selected. The washer limits the motion of the pinion outward along its rotational axis by defining a contact surface on
the concave side of the washer. When we select a TYPE = SINGLESIDED, a race surface is defined on the convex side for interfacing with the carrier mesh. The washer geometry and FE mesh size are also set within the washer menu, shown in Figure 14.31.

14.4.2.3 The Bevel Pinion TOOTH  Figure 14.32 shows the pinion tooth details, and Figure 14.33 shows the pinion tooth menu into which the data is entered.

14.4.2.4 The Bevel Pinion RIM  The rim is modeled within the RIM menu, with RIMTYPE set to SPHERICAL. The INSIDESPERHICALRADIUS, OUTSIDESPERHICALRADIUS, and INNERDIA inputs are shown in Figure 14.34. The rim is not offset in the radial or axial directions, and the finite element mesh is defined by NTHETA = 32, NAXIAL = 4, and NRADIAL = 4. The CIRCORDER = 4 and AXIALORDER = 2 matches the orders defined at the base surface of the bevel pinions.
Figure 14.32: The bevel pinion tooth details.
Figure 14.33: The bevel pinion TOOTH menu.
Figure 14.34: The bevel pinion rim.
14.4.3 The Half-Shafts

The left and right wheel half shafts form rotors 3 and 4. These are shown in Figures 14.37 and 14.38. Figures 14.35 and 14.36 show the menus for these two rotors. These two rotors each consist of one straight bevel pinion and three shafts.

The origins of both the rotors are located, for convenience, at the crossing point of the straight bevel pinion and gear axes. This is at \((X, Y, Z) = (0, 0, 64.296)\). So we have \(X\text{POSN}=0\), \(Y\text{POSN}=0\) and \(Z\text{POSN}=64.296\) for both the rotors. The left wheel half shaft (rotor 3) has its axis pointing along the +Z direction, so it has \(AX=0\), \(AY=0\) and \(AZ=1.0\). For the right wheel half shaft (rotor 4), the axis points along the -Z direction, so it has \(AZ=-1.0\).

Since we would like to specify the angular velocity for the left half and torque on the right half shaft, their TYPE is set to INPUT and OUTPUT respectively. We specify RPM = -1000 for the left half shaft and TORQUE = 2.5E6 Nmm for the right half shaft. Note that since the rotor axes for the two half shafts point in opposite directions, this means that they are actually rotating in opposite directions. In order to make them spin in the same direction, sign of the RPM value for the left half is negative. We have checked the ENABLESHAFTS and ENABLEBEVELS flags and set NSHAFTS to 3 and NBEVELS to 1.

We would like to hold the wheel end of the half shafts, so we check the UXCONSTRAINT, UYCONSTRAINT, UZCONSTRAINT, THETAXCONSTRAINT, THETAYCONSTRAINT and THETAZCONSTRAINT flags. UX, UY, UZ, THETAX and THETAY are all zero.

Figure 14.35: The rotor menu for the left wheel rotor.
Figure 14.36: The rotor menu for the right wheel rotor.
Figure 14.37: Rotor 3 shaft details.
Figure 14.38: Rotor 4 shaft details.
14.4.3.1 The Shafts  The shaft models for rotors 3 and 4 are very similar. They differ only in the lengths of a few segments on the first shaft. Shaft 1 for each rotor has six segments. The outside of segment 1 connects to shaft 2. The outer diameters of segments 3 and 5 form races where the rotors connect to stiffness bearings. The outer diameter of segment 2 is set to flexible constraint. This attaches the finite element model to the rotor reference frame. The wheel torque and any wheel forces act upon this segment.

The shaft 2 and 3 details are shown in Figure 14.39. The outer surfaces of shaft 2, segments 1 and 2 connect to the bevel gear base surface. The inner surface of segment 2 connects to shaft 1. ENABLEBACKINTERFACE is selected within the SHAFT menu for shaft 3 in order to connect the back surface of shaft 3 to the front side of the bevel gear surface.
**14.3.2 The Bevel Gear**  
The bevel gears on the two half shafts are identical, and are shown in Figure 14.40. Figures 14.41 and 14.42 show the menus into which the bevel gear details are entered.
Figure 14.41: The menu for a bevel gear of rotors 3 and 4.

Figure 14.42: The menu for a bevel gear of rotors 3 and 4.
14.4.4 The Stiffness Connectors

We add 5 new connectors to the model in this step. Connectors 7 through 10 are stiffness bearings that connect the half shafts to the carrier or to ground. Connectors 7 and 8 connect the half shaft rotors to ground and connectors 9 and 10 connect the half shafts to the carrier rotor. The 6th connector is the pinion stiffness bearing that connects the bevel pinion to the pin shaft. Stiffness bearings are theoretical spring stiffness connections between rotors. The stiffness values can be specified as linear stiffnesses in the x, y, or z directions or as torsional stiffnesses about the x or y (non-rotational) axis. The race positions of the half shaft connectors are shown in 14.4, and the stiffness values for these four connectors are provided in Table 14.5. The bevel pinion bearing CONNECTORS menu with inputs is shown in Figure 14.43.
Table 14.4: The half shaft connector origin and race positions.

<table>
<thead>
<tr>
<th>Connector</th>
<th>XPOS</th>
<th>YPOS</th>
<th>ZPOS</th>
<th>MEMBER1</th>
<th>AXPOS1</th>
<th>AXPOS2</th>
<th>DIA</th>
<th>MEMBER2</th>
<th>AXPOS1</th>
<th>AXPOS2</th>
<th>DIA</th>
</tr>
</thead>
<tbody>
<tr>
<td>7</td>
<td>0.00</td>
<td>0.00</td>
<td>1138.016</td>
<td>ROTOR3</td>
<td>-8.00</td>
<td>8.00</td>
<td>45.00</td>
<td>GROUND</td>
<td>-8.00</td>
<td>8.00</td>
<td>64.136</td>
</tr>
<tr>
<td>8</td>
<td>0.00</td>
<td>0.00</td>
<td>-918.984</td>
<td>ROTOR4</td>
<td>-8.00</td>
<td>8.00</td>
<td>45.00</td>
<td>GROUND</td>
<td>-8.00</td>
<td>8.00</td>
<td>64.136</td>
</tr>
<tr>
<td>9</td>
<td>0.00</td>
<td>0.00</td>
<td>167.016</td>
<td>ROTOR1</td>
<td>-23.00</td>
<td>23.00</td>
<td>62.614</td>
<td>ROTOR3</td>
<td>-23.00</td>
<td>23.00</td>
<td>45.00</td>
</tr>
<tr>
<td>10</td>
<td>0.00</td>
<td>0.00</td>
<td>-47.097</td>
<td>ROTOR1</td>
<td>-32.00</td>
<td>32.00</td>
<td>62.00</td>
<td>ROTOR4</td>
<td>-32.00</td>
<td>32.00</td>
<td>45.00</td>
</tr>
</tbody>
</table>
### Table 14.5: Half Shaft Bearing Stiffness Inputs

<table>
<thead>
<tr>
<th>ID</th>
<th>$k_x$</th>
<th>$k_y$</th>
<th>$k_z$</th>
<th>$k_{\theta_x}$</th>
<th>$k_{\theta_y}$</th>
<th>$k_{\theta_z}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>7</td>
<td>1.00E+05</td>
<td>1.00E+05</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>8</td>
<td>1.00E+05</td>
<td>1.00E+05</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>9</td>
<td>1.00E+05</td>
<td>1.00E+05</td>
<td>1.00E+09</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>10</td>
<td>1.00E+05</td>
<td>1.00E+05</td>
<td>1.00E+09</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
</tbody>
</table>
Figure 14.43: The CONNECTORS menu for the bevel pinion bearing.
14.4.5 The Bevel Pinion-Half Shaft Gear Pairs

The addition of the bevel planetary gears and two half shaft bevel gears requires two additional gear pairs. The PAIRS menu inputs for the bevel pinion-right wheel half shaft pair is shown in Figure 14.44. The left wheel half shaft bevel-bevel pinion pair is identical to the right, with just the IDROTOR input differing.
14.4.6 Using Spherical Carrier Races and Shafts to Model Side Bevel-Washer Contact

In this section, we demonstrate the use of spherical carrier races and spherical shafts to model a contact condition between the back of the side bevel gears and the carrier. A modified version of the Rear Axle example described to this point has been created and is available as a standalone model in the SAMPLES/RearAxleTraining/DifferentialCarrierSphericalRace directory on the product support website. In this section, we describe the important steps associated with modeling the spherical races and shafts. The remaining steps have been described previously in this example. Note, in order to model the spherical contact, the carrier and bevel gear/rim geometry have been modified. A schematic of the simplified model is provided in Figure 14.45.

14.4.6.1 The Carrier Rotor

The carrier rotor in the simplified model consists of the carrier and 3 shafts: the 2 spherical shafts (washers) and a shaft at the hypoid-carrier interface in order to constrain the rotor at the location where the torque flows in/out. The carrier inputs remain the same as the full model with the exception of the two races needed to attach the side bevel washer shafts.

The carrier RACE menu for one of the spherical carrier races is shown in Figure 14.47 and Figure 14.48 shows the menu inputs graphically. The AXIS direction is set to same in this case because the surface is in the same direction as the carrier rotor axis. An OFFSET=-24.72 positions the race origin so that it coincides with the sphere center. A positive SPHERICALRACE value of 74.846 is used so that the surface is concave when looking at the surface from the race origin. A negative spherical radius would be used in the case of a convex surface. The MINORANGLE and MAJORANGLE inputs define the inner and outer diameters of the race, measured as angles from the axis of rotation.

The spherical SHAFT and SEMGENT menus are shown in Figures 14.49 and 14.50, respectively. The shaft details for one of the two washer shafts are provided in the schematic in Figure 14.51. The shaft segment inputs include the thickness, inner and outer diameters, and front and back spherical radii. A positive spherical radius results in a convex surface, while a negative value results in a concave surface. The race interface is enabled within the SHAFT menu and the orders must match those defined within the carrier RACE menu. The contact surface is defined within the segment menu by turning on FRONTCONTACT or BACKCONTACT. In the case of the shaft shown in Figure 14.51, we set...
Figure 14.46: The carrier rotor menu.

ENABLEFRONTINTERFACE and BACKCONTACT to TRUE. The 2nd washer shaft is symmetric about the y axis to the shaft pictured.
Figure 14.48: The carrier races.
STEP 4: MODELING THE BEVEL PLANETARY SET W/BEARING, AND HALF SHAFT ROTORS W/STIFFNESS SUPPORT BEARINGS

Figure 14.49: The spherical SHAFT menu.

Figure 14.50: The spherical shaft SEGMENT menu.
Figure 14.51: The spherical shaft details.
14.4.6.2 The Bevel Rotors  The bevel rotors in this example consist of just the bevel tooth and two shafts that make up the rim. The schematic of the rotor shafts is shown in Figure 14.52 and the modified tooth menu inputs are provided in Figure 14.53. The spherical surface is modeled by setting DOFRONTCURVATURE to TRUE and entering the spherical radius into the FRONTSPHERICALRADIUS field. The second bevel gear rotor is symmetric about the y axis. Each bevel gear rotor is left unconstrained in all degrees of freedom since the tooth-tooth contact and spherical shaft contact sufficiently constrain the rotor in all degrees of freedom.
Figure 14.53: The bevel gear TOOTH geometry menu.
14.4.6.3 Contact Pressure on Spherical Surface  Figure 14.54 shows the contact pressure distribution on the spherical washer shaft surface resulting from the reaction with the spherical shaft of the side bevel gear.
14.5 Step 5: Importing the FE Housing Model

The FE housing model imported in this step is shown in Figure 14.55. Figure 14.56 shows the housing menu for the NASTRAN_EXTERNALFE housing TYPE. We constrain the housing reference frame degrees of freedom by setting UXCONSTRAINT, UYCONSTRAINT, UZCONSTRAINT, THETAXCONSTRAINT, and THETAYCONSTRAINT to TRUE. The name of the Nastran file is entered into the FILENAME field as rearaxle_housing.bdf.

NODETOLERANCE sets the tolerance used at the user defined race surface to find connecting nodes. MAXIMUMJOINTANGLE sets the maximum angle used to smooth joining shell elements. Joining elements forming angles greater than this amount are left as sharp edges. We use the recommended angle of 15 degrees in this example. For the sake of convenience, our system’s reference frame was made to coincide with the reference frame of the housing model. To do so, we set XSHIFT = YSHIFT = 0, and ZSHIFT = 44.016.

The RACE menu is shown in Figure 14.58. X, Y, and Z are the locations of the race origins in the reference frame of the CAD housing. In this case, housing reference frame is aligned with our global model reference frame, so the race coordinates are identical to the bearing coordinates. AX, AY, and AZ are the unit vectors that define the orientation of the positive race axis. AXPOSN1 and AXPOSN2 set the race length by defining the axial position of each edge from the race origin. The RACE menu inputs used for this example can be obtained from Figure 14.57.

14.5.1 Updated Connector Menu Inputs

We now connect the half shaft connectors to the housing instead of ground as was done in the previous step. Table 14.6 shows the updated connectors menu for the half shaft connectors. MEMBER2TYPE for connectors 7 and 8 is set as HOUSING and IDHOUSING is set to 1. The connector origins and race locations remain unchanged.
Figure 14.56: The housing menu.
Figure 14.57: Housing race locations and dimensions.
Figure 14.58: Housing race menu for race 1.
Table 14.6: The half shaft connector origin and race positions.

<table>
<thead>
<tr>
<th>Connector</th>
<th>XPOS</th>
<th>YPOS</th>
<th>ZPOS</th>
<th>MEMBER1</th>
<th>AXPOS1</th>
<th>AXPOS2</th>
<th>DIA</th>
<th>MEMBER2</th>
<th>AXPOS1</th>
<th>AXPOS2</th>
<th>DIA</th>
</tr>
</thead>
<tbody>
<tr>
<td>7</td>
<td>0.00</td>
<td>0.00</td>
<td>1138.016</td>
<td>ROTOR3</td>
<td>-8.00</td>
<td>8.00</td>
<td>45.00</td>
<td>HOUSING1</td>
<td>-8.00</td>
<td>8.00</td>
<td>64.136</td>
</tr>
<tr>
<td>8</td>
<td>0.00</td>
<td>0.00</td>
<td>-918.984</td>
<td>ROTOR4</td>
<td>-8.00</td>
<td>8.00</td>
<td>45.00</td>
<td>HOUSING1</td>
<td>-8.00</td>
<td>8.00</td>
<td>64.136</td>
</tr>
<tr>
<td>9</td>
<td>0.00</td>
<td>0.00</td>
<td>167.016</td>
<td>ROTOR1</td>
<td>-23.00</td>
<td>23.00</td>
<td>62.614</td>
<td>ROTOR3</td>
<td>-23.00</td>
<td>23.00</td>
<td>45.00</td>
</tr>
<tr>
<td>10</td>
<td>0.00</td>
<td>0.00</td>
<td>-47.097</td>
<td>ROTOR1</td>
<td>-32.00</td>
<td>32.00</td>
<td>62.00</td>
<td>ROTOR4</td>
<td>-32.00</td>
<td>32.00</td>
<td>45.00</td>
</tr>
</tbody>
</table>
14.6 The Analysis Setup

Figure 14.59 shows the analysis setup menu for analyzing a single time step of the system. Only one time step is being analyzed, so NTIMESTEPS is 1. The results of the file are stored in the file with name POSTFILENAME=postproc1ts.dat. Note, to run the analysis with the roller bearings, the abovementioned file name is POSTFILENAME=postproc_RollerBearings1ts.dat.
## Figure 14.59: The analysis setup menu

![Analysis Setup Menu](image_url)

### MultyX Setup

- **EXIT**
- **QUIT**

### RANGE

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>CONSERVEMEMORY</td>
<td>Checked</td>
</tr>
<tr>
<td>INCLUDE EDGE CONTACT</td>
<td>Checked</td>
</tr>
<tr>
<td>ZERONLLIAL</td>
<td>Checked</td>
</tr>
<tr>
<td>INITIAL TIME</td>
<td>-0.0001428600</td>
</tr>
<tr>
<td>NRANGES</td>
<td>1</td>
</tr>
<tr>
<td>SAVEPERIODICALLY</td>
<td></td>
</tr>
<tr>
<td>OUTPUTRESTART</td>
<td></td>
</tr>
<tr>
<td>POSTPROCWRITE</td>
<td>Checked</td>
</tr>
<tr>
<td>SPLITPOSTPROCFILE</td>
<td>Checked</td>
</tr>
<tr>
<td>POSTFILENAME</td>
<td>postproc1ts.dat</td>
</tr>
<tr>
<td>FREEMODETHRESHOLD</td>
<td>1.0000000000e-08</td>
</tr>
<tr>
<td>CALYXVERBOSITY</td>
<td>5</td>
</tr>
</tbody>
</table>
14.7 Results

The analysis of the full system on a 1700 MHz Intel Pentium took eight hours to formulate and decompose all stiffness matrices. It took a further 3 hours for each time step analyzed.

Figures 14.60 through 14.63 show stress contour and contact pattern results for the analysis of the full system after step 5.
Figure 14.61: Stress contours on the carrier with roller bearings.
Figure 14.62: Stress contours on the housing with roller bearings.
Figure 14.63: Stress contours on the housing with roller bearings.
### Table 14.7: E, P, G, $\alpha$ Deflection Values

<table>
<thead>
<tr>
<th>Time</th>
<th>E (mm)</th>
<th>P (mm)</th>
<th>G (mm)</th>
<th>$\alpha$ (rad)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.429E-004</td>
<td>-2.087E-001</td>
<td>1.755E-002</td>
<td>1.343E-001</td>
<td>1.460E-004</td>
</tr>
<tr>
<td>2.857E-004</td>
<td>-2.076E-001</td>
<td>2.059E-002</td>
<td>1.358E-001</td>
<td>1.616E-004</td>
</tr>
<tr>
<td>4.286E-004</td>
<td>-2.047E-001</td>
<td>1.030E-002</td>
<td>1.326E-001</td>
<td>1.899E-004</td>
</tr>
<tr>
<td>5.714E-004</td>
<td>-2.075E-001</td>
<td>6.870E-003</td>
<td>1.320E-001</td>
<td>1.938E-004</td>
</tr>
<tr>
<td>7.143E-004</td>
<td>-2.154E-001</td>
<td>4.805E-003</td>
<td>1.313E-001</td>
<td>1.909E-004</td>
</tr>
<tr>
<td>8.572E-004</td>
<td>-2.223E-001</td>
<td>-1.320E-004</td>
<td>1.314E-001</td>
<td>1.886E-004</td>
</tr>
<tr>
<td>1.000E-003</td>
<td>-2.337E-001</td>
<td>2.522E-002</td>
<td>1.360E-001</td>
<td>1.458E-004</td>
</tr>
<tr>
<td>1.143E-003</td>
<td>-2.170E-001</td>
<td>2.663E-002</td>
<td>1.320E-001</td>
<td>1.241E-004</td>
</tr>
<tr>
<td>1.286E-003</td>
<td>-2.187E-001</td>
<td>3.268E-002</td>
<td>1.326E-001</td>
<td>1.295E-004</td>
</tr>
<tr>
<td>1.429E-003</td>
<td>-2.150E-001</td>
<td>2.123E-002</td>
<td>1.302E-001</td>
<td>1.609E-004</td>
</tr>
<tr>
<td>1.571E-003</td>
<td>-2.097E-001</td>
<td>9.195E-003</td>
<td>1.270E-001</td>
<td>1.981E-004</td>
</tr>
</tbody>
</table>

### 14.7.1 EPGAlpha Results and Explanation

The E, P, G, $\alpha$ (V, H, G, $\Delta\sigma$) deflection values are generated by *Transmission3D* upon completion of the analysis. The values are located in a file called `EPGALPHA.DAT`, which can be found in the `calyxtmp` folder in the working directory. The file is organized left-to-right in the following order: time, E, P, G, $\alpha$ and top-to-bottom by the time steps for which the analysis is run. These deflection values can be used in the *Gleason-CageWin* and *Klingelnberg Kimos* design software for the purpose of re-designing the hypoid gear pair. Table 14.7 displays the E, P, G, $\alpha$ (rad) results for the Rear Axle model.

The E, P, G, $\alpha$ values are calculated by first measuring the 3-dimensional deflection at the nodes on the base surface of the loaded teeth. This is done for both the pinion and gear, separately. Next, a least squares regression of the 6 rigid body motion components is calculated to best fit the measured deflection (12 components total - 6 gear, 6 pinion). The 12 rigid body components are then converted into 6 relative motion components. Since rotation of the pinion and gear about each of their respective axis does not contribute to the deflection, we ignore these components. The remaining 4 components are the E, P, G, and $\alpha$ values. The sign convention for E, P, G, and $\alpha$ for both left and right-handed gears are described in Figures 14.64 through 14.67.
Figure 14.64: The E,P,G,Alpha sign convention for a left-handed gear.

Figure 14.65: The E,P,G,Alpha sign convention for a left-handed gear.
Figure 14.66: The E,P,G,Alpha sign convention for a right-handed gear.

Figure 14.67: The E,P,G,Alpha sign convention for a right-handed gear.
In this chapter, we describe the process for running a dynamic analysis of the Rear Axle Model in Transmission3D in order to obtain the housing structure velocity frequency response for use as the boundary condition in a Courstyx acoustic simulation of the housing.

15.0.1 Transmission3D Dynamic Analysis Setup

For this example, we take the rear axle model from the previous chapter and modify the analysis parameters in order to analyze the dynamics that result from excitation of a gear pair. The dominant source of excitation in the model is the hypoid gear tooth mesh so we choose analysis parameters that capture the mesh and shaft frequencies and related harmonics of this pair. We do so by selecting a sampling frequency, or time step increment, that is divisible by the mesh frequency. The shaft frequency is naturally divisible by the mesh frequency, so selecting a time step based upon the mesh frequency will capture both shaft and mesh frequencies. The mesh cycle time, or inverse of mesh frequency, is calculated using Equation 15.1. The resulting mesh cycle time for the hypoid gear pair is 1.42857E-03 seconds, which is equivalent to a 700 Hz mesh frequency.

\[
\text{MeshCycleTime}(s) = \frac{\omega_{\text{pinion}} \cdot N\text{TEETH}_{\text{pinion}}}{2\pi} = \frac{1}{f_{\text{mesh}}} \quad (15.1)
\]

15.0.1.1 Determination of DELTATIME and NTIMESTEPS Parameters

The frequency range of the dynamic response is dependent upon the time increment (DELTATIME) and the total number of time steps (NTIMESTEPS) as described by Equations 15.2 and 15.3. When selecting the DELTATIME parameter three things must be considered: 1) the mesh cycle time of the hypoid gear pair must be a multiple of the DELTATIME, 2) the DELTATIME selected must be sufficiently small to capture frequencies up to the desired maximum frequency as described by Equation 15.2, and 3) at least 40 time steps per mesh cycle are recommended for numerical stability of the model. Using 40 time steps per mesh cycle, or DELTATIME = 3.5714285E-05 s, the maximum frequency is \( f_{\text{mesh}} \cdot 40 \) = 14 kHz. If the resulting maximum frequency is below the desired frequency, a value larger than 40 may be used to increase the

By Advanced Numerical Solutions Copyright © 2021 Advanced Numerical Solutions, LLC
maximum frequency. Increasing the value to 60 steps per mesh cycle, we obtain a max frequency of \( \frac{f_{\text{mesh}} \times 60}{2} = 21 \) kHz, which covers the maximum audible frequency of 20 kHz. The corresponding DELTATIME parameter changes to \( \frac{\text{time}_{\text{Mesh Cycle}}}{60} = 2.38095 \times 10^{-5} \) seconds. The number of analysis time steps (NTIMESTEPS) and resulting minimum, or delta, frequency can then be calculated using Equation 15.3.

Both the DELTATIME and NTIMESTEPS parameters are entered into the SETUP > RANGE menu. We recommend running a dynamic analysis with a minimum of 3 ranges: one single time step quasi-static range, a transient dynamic range, and at least one steady state range for which the dynamic results are generated. The single quasi-static step provides a solution that is used as the initial condition for the dynamic analysis, thus reducing transient effects and solution time. The number of time steps for the transient range should be selected by running an initial analysis and plotting the time response of the two hypoid gear rotors to observe the length of the transients. Figures 15.1 and 15.2 show the time responses of the \( \theta_z \) component of the hypoid pinion and gear rotors, respectively, for a preliminary analysis run with a 1 time step quasistatic range and a 10,000 time step Newmark dynamic range. The transient portion of each time response reach steady state within 4,500 time steps, so we set NTIMESTEPS for range 2 to 4,500. Selecting 12,000 steady state time steps for range 3 gives us a minimum frequency of 3.5 Hz from Equation 15.3. The resulting frequency range, 3.5 Hz to 21 kHz, is equivalent to \( \frac{f_{\text{mesh}}}{200} < f < 30 \times f_{\text{mesh}} \) meaning the analysis can capture 200 subharmonic and 30 harmonic frequencies of the hypoid tooth mesh frequency.

\[
\text{MaxFrequency}(Hz) = \frac{1}{2 \times \text{DELTATIME}} \tag{15.2}
\]

\[
\text{MinFrequency}(Hz) = \frac{1}{\text{DELTATIME} \times \text{NTIMESTEPS}_\text{Steady State}} \tag{15.3}
\]
Figure 15.2: The hypoid gear dynamic time response $\theta_z$ component.
15.0.1.2 OP2 File Output Setup  

Transmission3D has the ability to generate Nastran OP2 results files consisting of displacement, velocity, and/or acceleration data in the frequency domain. For this example, we would like to load the housing velocity frequency response directly into the Coustyx acoustic model so that we can avoid the time consuming process of converting between time and frequency domain in Coustyx.

The OP2 results file is generated for the steady state range by selecting the WRITEOP2FILES box within the RANGE menu with IRANGE = 3. Figure 15.3 shows the RANGE menu parameters set to generate the OP2 file. The contents of the results file is set in the OP2FILES submenu of the main menu. For the acoustic analysis, the OP2 file should just contain the velocity data, so we select only the velocity box. Figure 15.4 shows the OP2FILES menu inputs. The FREQUENCYTABLE submenu is where we define the frequency range for the housing displacement results that are written to the OP2 file. We load the frequency data from a script file called FrequencyTable.txt. This file is included with the model files for this example and includes frequencies from 175 Hz to 7 kHz in increments of 175 Hz. The frequencies are of subharmonics and harmonics of the hypoid gear tooth mesh that is the main source of excitation. To load the frequency data, we go to the FREQUENCYTABLE submenu of the OP2FILES menu, select EXECUTESCRIPT from the home panel in the Guide model editor, and select the FrequencyTable.txt file from the model working directory. Execution of the dynamic analysis generates the OP2 file with the filename HOUSING_RESULTS.OP2 containing the housing velocity data for frequencies in the range of \( \frac{f_{\text{mesh}}}{4} < f < 10 \times f_{\text{mesh}} \).
Figure 15.4: OP2 file setup from the OP2FILES menu.

![Image of OP2 file setup](image-url)
15.0.1.3 Final Analysis Run and Results

At this stage the model is setup for the final dynamic run. During the analysis deflection results files for the model rotors, bearings, and housings to the calyxtmp folder with the filenames ending in *RES.DAT. We plot the full time response and the FFT of the steady state frequency response utilizing a Python script name PlotT3DDynamics.py. This script is available within the model files for this example and requires the user enter the full path to the directory where the deflection results files are located, the desired filenames and deflection components to plot, and number of steady state time steps for the FFT.
Figure 15.5: Importing the structure into Coustyx from the OP2 results file.

15.0.2 Coustyx Acoustic Model Setup

We utilize the Coustyx software to perform a multi-domain acoustic simulation on the housing structure, applying the housing structure nodal velocity data obtained from the Transmission3D dynamic analysis as the boundary condition for the acoustic model. To create the Coustyx model, we select File > New Model and the select multi-domain model and the desired units system from the popup window. The structure mesh from the OP2 file is imported by right-clicking on the Structures item within the menu tree and selecting Import Mesh > Nastran OP2 Data as shown in Figure 15.5.

Next, we need to create a boundary element (BE) mesh on which we can apply a structural velocity boundary condition (BC). A BE mesh is created in Coustyx by first creating a skin of the enclosed structure and then converting the skin into a BE mesh. In order to create a skin in Coustyx, the mesh to be skinned should be fully enclosed, and the mesh element size should be consistent. For this example, we create a shrink wrap mesh of the housing structure using an external FE preprocessing software package in order to obtain an enclosed and uniform mesh. We import the skin mesh as a 2nd structure by selecting Import Mesh > Nastran Bulk Data File, as shown in Figure 15.6. The shrink wrap mesh file, Loose_shrinkwrap_5mm.bdf, is included in the example files in the /SAMPLES/Rear Axle Dynamics/ directory.

We can now skin the imported shrink wrap mesh and convert it to a BE mesh. We do this by first opening the mesh structure in the 3D view window by right clicking the structure in the menu tree and selecting Open. Within the 3D view, we select one element by holding SHIFT + left-clicking on the 3d housing structure. With one element selected, we click the Create Skin button within the Skin tab. After the skinning process is complete, the boundary element mesh is created by clicking the Create Mesh From Skin button. A new BE mesh should now appear within the Direct BE Meshes menu tree folder. Figure 15.7 shows the Skin panel used to create the skin and BE mesh.
Figure 15.6: Importing the external shrink wrap mesh in Coustyx.

Figure 15.7: Creating the skin and BE mesh.
The structure velocity data generated from the Transmission3D dynamic analysis is applied as a BC to the BE mesh by right-clicking on the Boundary Conditions folder in the menu tree and selecting New. The New Boundary Condition dialog box, shown in Figure 15.8, is used to define the structure velocity boundary condition parameters. We select the structure name from the dropdown box, and then define the interpolation parameters that are used to interpolated between the BE mesh nodes and the nodes of the structure mesh for which we have the normal structure velocity data. The number of interpolating points is set to 4 by default and can be adjusted as needed. Unselecting the Choose Default Options box, we can set the Maximum Search Distance based upon the size of the two meshes. For this example, we have a BE mesh with 5 mm elements, and a structure mesh of varying element size between 1 mm and 10 mm, so we select a value of 5.

The acoustic analysis parameters are set by right-clicking the Analysis Sequences folder in the menu tree and selecting New. We select the Acoustic Response: Solve and Post Process sequence type from the dropdown menu to execute a new acoustic analysis. The Solver Controls tab, shown in Figure 15.10, is used to set the solver related parameters. Within the Frequency Ranges tab (Figure ??) we define the analysis frequencies. We can enter these manually, or load the frequencies defined in the Transmission3D OP2FILES menu. The Just In-Time Loading tab, shown in Figure 15.13, allows us to link the Transmission3D velocity frequency response data for dynamic loading during run-time. If the frequencies defined in the Frequency Ranges tab does not match the frequency loaded from the OP2 file, the two closest frequencies from the OP2 file are loaded and interpolated to estimate the velocity magnitude at the specified analysis frequency.

The analysis sequence is run by highlighting the sequence in the menu tree and selecting the run button. The output parameters for this example are set to produce a sensor results file, sound power results file, and an iGlass 3D results viewer file. The iGlass 3D contour view of the surface pressure on the housing is shown in Figure 15.14.
Figure 15.9: Setting the structure velocity boundary condition.

Figure 15.10: Setting up the analysis sequence solver controls.
Figure 15.11: Loading the analysis sequence frequencies from the structure response.

Figure 15.12: The analysis sequence frequency table.
Figure 15.13: Setting up just in time loading of housing frequency response.
Figure 15.14: The 3D sound pressure contour iGlass results.
CHAPTER 16

CALCULATION OF LUBRICANT FILM THICKNESS & FLASH TEMPERATURE

16.1 Introduction

This example demonstrates the use of Transmission3D to calculate flash temperature and lubricant film thickness for spur gears. The sample gears of the ‘ISO TR 15144-1:2010, Calculation of micropitting load capacity of cylindrical spur and helical Gears’ is modeled to demonstrate and validate Transmission3D calculations. The inputs for this example can be loaded from session file FilmThickness.ses in the subdirectory SAMPLES/FilmThickness.

16.2 Model

This is a simple gear pair system with two rotors. The rotors are held in position by constraining all the degrees of freedom of the rotor reference frame.

16.3 Calculating Flash temperature and Film Thickness

The Transmission3D model is analyzed for one mesh cycle. The applied torque on the gear is 1878 Nm at 3000 RPM. The results are extracted from the PATTERN sub menu under post-processing. The Transmission3D shows a 2D distribution plot of flash temperature and film thickness in graphics window as shown in Figure 16.3 and 16.5. The results are also written to the output file. Figures 16.4 and 16.6 show the resulting patterns when the AUTOCOMPUTEMU option is selected.

16.4 Validation of Results

In the standard, the parameters are calculated at five different positions on the path of contact. The position on the path of contact are mapped on to the corresponding surface profile parameter,s in Transmission3D. The results from
Figure 16.1: Gear Pair Model.
Table 16.1: Gear Data

<table>
<thead>
<tr>
<th></th>
<th>Gear1</th>
<th>Gear2</th>
</tr>
</thead>
<tbody>
<tr>
<td>No. of Teeth</td>
<td>18</td>
<td>18</td>
</tr>
<tr>
<td>Normal Module</td>
<td>10.93</td>
<td></td>
</tr>
<tr>
<td>Normal Pressure Angle</td>
<td>20 deg</td>
<td></td>
</tr>
<tr>
<td>Normal Tooth Thickness</td>
<td>17.00 mm</td>
<td>17.00mm</td>
</tr>
<tr>
<td>Face Width</td>
<td>21.4 mm</td>
<td></td>
</tr>
<tr>
<td>Rack Tip Radius</td>
<td>0.1 mm</td>
<td>0.1 mm</td>
</tr>
<tr>
<td>Outside Diameter</td>
<td>221.4 mm</td>
<td>221.4 mm</td>
</tr>
<tr>
<td>Root Diameter</td>
<td>170 mm</td>
<td>170 mm</td>
</tr>
<tr>
<td>Inner Diameter</td>
<td>120 mm</td>
<td>120 mm</td>
</tr>
<tr>
<td>Young’s Modulus</td>
<td>2.06E5 N/mm²</td>
<td></td>
</tr>
<tr>
<td>Density</td>
<td>7.8e-9 Ns²/mm⁴</td>
<td></td>
</tr>
<tr>
<td>Template</td>
<td>FINEROOT.TPL</td>
<td></td>
</tr>
</tbody>
</table>
Figure 16.3: Flash Temperature Distribution with Friction Coefficient = 0.048
Figure 16.4: Flash Temperature Distribution w/Autocomputed Friction Coefficient
Figure 16.5: Film Thickness Distribution w/Friction Coefficient = 0.048
Figure 16.6: Film Thickness Distribution w/Autocomputed Friction Coefficient
### Table 16.2: Pattern Menu Material/Fluid Properties

<table>
<thead>
<tr>
<th>Parameter Name</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Specific Heat, CP, SI</td>
<td>Amount of heat required to change the temperature by a given amount per unit mass of the gear material. Units are SI (J/kg − K)</td>
</tr>
<tr>
<td>Specific Heat Conductivity, LAMBDA_SI</td>
<td>The specific heat conductivity of the gear material.</td>
</tr>
<tr>
<td>Pressure Viscosity Coefficient, ALPHA_38_SI</td>
<td>The pressure viscosity coefficient at at 38°C.</td>
</tr>
<tr>
<td>Kinematic Viscosity of Lubricant, ETA_40_METRIC</td>
<td>Kinematic viscosity of the lubricant at 40°C.</td>
</tr>
<tr>
<td>Kinematic Viscosity of Lubricant, ETA_100_METRIC</td>
<td>Kinematic viscosity of the lubricant at 100°C.</td>
</tr>
<tr>
<td>Density of Lubricant, DENSITY_15_SI</td>
<td>Density of lubricant at 15°C. Units are SI (kg/m³)</td>
</tr>
<tr>
<td>Surface Bulk Temperature, BULKTEMP_SI</td>
<td>The equilibrium temperature of the surface of the gear teeth before entering contact zone. Units are SI (°C)</td>
</tr>
<tr>
<td>Surface Roughness, RA_MICRONS</td>
<td>The effective mean roughness value. Units in microns.</td>
</tr>
</tbody>
</table>

The standard are compared against the Transmission3D calculations. The results along the mid-facewidth of the gears in Transmission3D are used for comparison. The parameters compared are rolling velocity, sliding velocity, contact pressure, relative normal curvature, flash temperature, and film thickness. The results show the close comparison of the basic parameters, rolling velocity, sliding velocity, and radius of curvatures. The variation seen in contact pressure values due to the difference in the load sharing calculation. This effect is carried down into the flash temperature and the film thickness values.

The table of values from ISO standard and Transmission3D are given in the excel file FilmThicknessResults.xlsx included in the examples folder SAMPLES/FilmThickness. NOTE: The ISO standard uses the 0.048 value for the coefficient of friction for its calculations.
Figure 16.7: Comparison of Results.
Figure 16.8: Comparison of Results.
Figure 16.9: Comparison of Results.
Figure 16.10: Comparison of Results.
17 Overview

In this example, we demonstrate the dynamic condensation process for a simply supported plate model using both 
Abaqus CAE and Hypermesh/Optistruct. We then verify the validity of the condensed structures by calculating their 
natural frequencies using Transmission3D and comparing them to the theoretical modal frequencies for a simply sup-
ported plate. The model files required to exercise this example can be found in the SAMPLES/DynamicCondensationThinPlate 
directory on the Ansol technical support website (http://techsupport.ansol.com).

17.1.1 Dynamic Condensation of a Structure with Hypermesh/Optistruct

To demonstrate the condensation of a structure in Hypermesh, we first model a 500 mm x 500 mm surface and mesh it 
using shell elements. The grid size is 25 elements x 25 in Figure 17.1. We assign the MAT1 material card containing 
the Transmission3D default steel material properties (E=2.06e+05, ρ=7.8e-09, ν=0.3) to a PSHELL property card. 
The dynamic condensation of the model is performed with Optistruct by following the steps below.

1. Create a load collector by selecting 'Collectors > Create > LoadCollectors' from the drop-down menu. We name 
the collector 'ASET' and set the card image to 'No Card Image'. We then apply ASET constraints by selecting 
'Analysis > Constraints' from the panel at the bottom of the Hypermesh application window. The constraint 
panel is shown in Figure 17.2. Ensure the ASET load collector is active when creating the constraints so that 
they are assigned to the correct load collector. An active load collector is signified with its name in bold font in 
the model tree. The ASET constraints define the interface nodes to which the structure will condense down to. 
Selecting all 6 DOF means we will end up with 24 total dofs in addition to the number of desired modal dofs.

2. Create a second load collector from the drop-down menu and name it 'SPC'. Set the card image to 'No Card 
Image'. To model a simply supported constraint on the two edges perpendicular to the x-axis, constrain all but 
the THETAY rotational dof by leaving the ‘dof 5’ box unselected as shown in Figure 17.3. Similarly, with the 
SPC card still active, we constrain all but the THETAX dof on the two edges perpendicular to the y-axis by 
selecting all but the ‘dof 4’ box.
3. Set the analysis method and associated parameters with the CMSMETH load collector. This load collector is also created from the Collectors drop-down menu. Selecting the 'CMSMETH' card image will bring up the card inputs shown in Figure 17.1. We set the type to 'Structure Only' and method to 'CBN' (Craig-Bampton). The number of desired modes to output is set to 100 and the solver to 'Lanczos' (Lanczos Algorithm). Lastly, we set the SPID to the node number at which we would like to begin the numbering the modes. The SPID can be any number that is not currently being used to identify a GRID point (node).

4. Create the control cards for the analysis inputs/outputs. For dynamic condensation, we use the PARAM, GLOBAL_CASE_CONTROL, and SCREEN cards. To setup the cards, we select Analysis > Control Cards from the Hypermesh panel and select the card name. The inputs for each card are shown in Figure 17.1. The PARAM>EXTOUT>DMIGPCH card generates the punch file (.pch) containing the condensed mass and stiffness matrices and modal dofs. Note: if an acoustic analysis in Coustyx is going to be performed, the PARAM>EXCOUT>3 card must also be set, in order to generate the .DOFT and .OUT4 files. The GLOBAL_CASE_CONTROL card points to the CMSMETH load collector for the analysis parameters, as well as the SPC card for the constrained nodes. The SCREEN card outputs the analysis log to the analysis information window that appears when the analysis is started.

5. Start the analysis by selecting 'Analysis > Optistruct' from the panel menu. Make sure the analysis options are set as shown in Figure 17.4 and click 'Optistruct' to begin the condensation.
Figure 17.2: Application of the ASET nodal constraints.
Figure 17.3: Application of the SPC nodal constraints.

Figure 17.4: Optistruct analysis panel menu.
Figure 17.5: The HOUSING menu inputs for the condensed housing type.

Table 17.1: T3D NODE menu inputs.

<table>
<thead>
<tr>
<th>INODE</th>
<th>1</th>
<th>2</th>
<th>3</th>
<th>4</th>
</tr>
</thead>
<tbody>
<tr>
<td>NODEID</td>
<td>162</td>
<td>261</td>
<td>494</td>
<td>675</td>
</tr>
<tr>
<td>XNODE</td>
<td>-110.0</td>
<td>110.0</td>
<td>-110.0</td>
<td>110.0</td>
</tr>
<tr>
<td>YNODE</td>
<td>110.0</td>
<td>110.0</td>
<td>-110.0</td>
<td>-110.0</td>
</tr>
<tr>
<td>ZNODE</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>CONSTRAIN</td>
<td>NO</td>
<td>NO</td>
<td>NO</td>
<td>NO</td>
</tr>
</tbody>
</table>

Table 17.2: Transmission3D vs Theoretical Natural Frequencies - Optistruct.

<table>
<thead>
<tr>
<th>Method</th>
<th>Mode</th>
<th>1</th>
<th>2</th>
<th>3</th>
<th>4</th>
<th>5</th>
<th>6</th>
<th>7</th>
<th>8</th>
<th>9</th>
</tr>
</thead>
<tbody>
<tr>
<td>Hypermesh/Optistruct</td>
<td>1</td>
<td>195.58</td>
<td>487.98</td>
<td>487.98</td>
<td>776.55</td>
<td>974.22</td>
<td>974.22</td>
<td>1256.40</td>
<td>1256.40</td>
<td>1652.60</td>
</tr>
<tr>
<td>Theoretical</td>
<td>2</td>
<td>195.40</td>
<td>488.60</td>
<td>488.60</td>
<td>781.70</td>
<td>977.10</td>
<td>977.10</td>
<td>1270.30</td>
<td>1270.30</td>
<td>1758.80</td>
</tr>
<tr>
<td>% Difference</td>
<td>3</td>
<td>.092</td>
<td>.127</td>
<td>.127</td>
<td>.663</td>
<td>.296</td>
<td>.296</td>
<td>1.106</td>
<td>1.106</td>
<td>6.426</td>
</tr>
</tbody>
</table>

17.1.1.1 Calculating Modal Frequencies of Condensed Structure from Transmission3D

The condensed plate model is imported into Transmission3D by setting the housing TYPE to NASTRAN_CONDENSED within the HOUSING menu as shown in Figure 17.5. The interface node IDs and coordinates are entered into the NODE submenu as shown in Table 17.3. For this example, we set the rotor 1 TYPE to INACTIVE since we only need the model to pass the generate phase. We then enter reasonable values into the rotor’s SHAFT and SEGMENT menus such that the model passes the logical checks performed by Multyx (shaft segment OD > ID, etc.). During model generation Multyx reads the condensed structure file and converts it into a Matlab external structure file (extstruct.m) containing the plate mass and stiffness matrices. Copying this file from the calyxtmp folder into the same directory as the CalcNaturalFreq.m program included with the model files allows us to calculate the natural frequencies of the plate. We do so by taking the square root of the eigenvalues of the mass and stiffness matrices from the file. The calculated frequencies are shown in Figure 17.4 and compared to the theoretical modal frequencies of the plate.
17.1.2 Model Setup and Substructuring Analysis with Abaqus

In this section we discuss the dynamic condensation process using Abaqus CAE. To begin, we create the plate and mesh it so that it is identical to the Hypermesh model described in the previous section. We then define the material using identical properties to those of the Hypermesh model. With Abaqus, a ‘section’ is created, material properties are assigned to the section, and the section is assigned to the mesh. The Craig-Bampton dynamic condensation process in Abaqus requires two analysis steps: 1) a frequency analysis of the model with the interface nodes fully constrained and 2) substructure generation with the interface nodes’ dofs retained. The procedure for setting up the two analysis steps is detailed below:

1. First, we define any boundary conditions that are applicable to all analysis steps under the 'Initial' tab by selecting 'BCs > Create'. For this example, we define the simply supported edge constraints here. To do so, we create a 'mechanical displacement/rotation constraint', select the set of nodes on the two edges perpendicular to the x-axis, and constrain all dofs except for rotation about the x-axis as shown in Figure 17.6. We apply the same conditions to the two edges perpendicular to the y-axis by constraining all but the URY dof.

2. Next, we create the frequency analysis step by right-clicking on the 'Steps' heading in the model tree and selecting 'Create'. We set the solver type to 'Lanczos', number of desired eigenvalues/modes to 100, and the remaining analysis parameters as shown in Figure 17.7. Clicking 'OK' creates a new analysis step titled 'Step-1' in the model tree. Within the 'Step-1' heading, we right-click on the 'BCs' tab heading and select 'Create'. In the dialogue box, we select 'Mechanical > Symmetry/Antisymmetry/Ecastre' and then 'Ecastre' to constrain all degrees of freedom as shown in Figure 17.8. When prompted, we select the set of interface nodes to constrain during the frequency analysis step.

3. The substructure generation analysis step is created similarly to the frequency analysis step in step 2. When the dialogue box appears, we select the options shown in Figure 17.9. The boundary condition for this analysis step is created by selecting the 'Mechanical > Retained nodal dofs' type and selecting the 4 nodes and degrees of freedom shown in Figure 17.10. This sets the 6 degrees of freedom at each of these 4 nodes as retained dofs, so we end up with 124 total degrees of freedom (6x4=24 + 100 retained modes = 124 dof).

4. A job is created by right clicking on 'Jobs', selecting 'Create' and using the settings shown in Figure 17.11. To start the analysis we right-click on 'Job-1' and select 'Submit'. The output files, are located within the temp folder defined in the Abaqus user preferences. We also must add a command line to the input file in order to output a condensed stiffness matrix file. We do so by selecting 'Model > Edit Keywords' from the dropdown menu, and typing the last two command lines in Figure 17.12. If doing an acoustic analysis with Coustyx, we must also add the commands for generating the recovery matrix. The condensed massstiffness matrix and recovery matrix are output to two files titled massstiffness.mtx and recovery.mtx, respectively.
Figure 17.6: The initial load step boundary conditions.

Figure 17.7: Creating the frequency analysis step.
Figure 17.8: The frequency analysis step boundary condition.

Figure 17.9: Creating the substructure generation step.
Figure 17.10: The substructure generation step boundary condition.
Figure 17.11: Creating the job for the analysis.
Figure 17.12: Editing the keywords to create condensed massstiffness and recovery matrix output files.
Table 17.3: T3D NODE menu inputs.

<table>
<thead>
<tr>
<th>INODE</th>
<th>1</th>
<th>2</th>
<th>3</th>
<th>4</th>
</tr>
</thead>
<tbody>
<tr>
<td>NODEID</td>
<td>162</td>
<td>261</td>
<td>494</td>
<td>675</td>
</tr>
<tr>
<td>XNODE</td>
<td>-110.0</td>
<td>110.0</td>
<td>-110.0</td>
<td>110.0</td>
</tr>
<tr>
<td>YNODE</td>
<td>110.0</td>
<td>110.0</td>
<td>-110.0</td>
<td>-110.0</td>
</tr>
<tr>
<td>ZNODE</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>CONSTRAN</td>
<td>NO</td>
<td>NO</td>
<td>NO</td>
<td>NO</td>
</tr>
</tbody>
</table>

Table 17.4: Transmission3D vs Theoretical Natural Frequencies - Abaqus.

<table>
<thead>
<tr>
<th>Method</th>
<th>Mode</th>
<th>1</th>
<th>2</th>
<th>3</th>
<th>4</th>
<th>5</th>
<th>6</th>
<th>7</th>
<th>8</th>
<th>9</th>
</tr>
</thead>
<tbody>
<tr>
<td>Abaqus</td>
<td>1</td>
<td>197.29</td>
<td>494.31</td>
<td>494.31</td>
<td>788.93</td>
<td>994.29</td>
<td>994.30</td>
<td>1284.90</td>
<td>1284.90</td>
<td>1704.70</td>
</tr>
<tr>
<td>Theoretical</td>
<td>2</td>
<td>195.40</td>
<td>488.60</td>
<td>488.60</td>
<td>781.70</td>
<td>977.10</td>
<td>977.10</td>
<td>1270.30</td>
<td>1270.30</td>
<td>1758.80</td>
</tr>
<tr>
<td>% Difference</td>
<td>3</td>
<td>.093</td>
<td>1.16</td>
<td>1.16</td>
<td>.916</td>
<td>1.73</td>
<td>1.73</td>
<td>1.14</td>
<td>1.14</td>
<td>3.17</td>
</tr>
</tbody>
</table>

17.1.2.1 Comparison with Theoretical Modal Frequencies We set up the Transmission3D model of the Abaqus condensed plate model in the same way as we did for the Optistruct model described in section 17.1.1.1, with the exception of the filename. The NODE menu inputs also change slightly due to the difference in node numbering between Optistruct and Abaqus. The updated NODE menu inputs are provided in Table 17.3. The remaining procedure for obtaining the modal frequencies from Transmission3D is identical. The frequency values obtained from Transmission3D using the Abaqus condensed plate model are compared to the theoretical frequency values in Table 17.4.
This example serves as a short tutorial on setting up and running a dynamic analysis with FE probes, and plotting the response of each probe’s output in the time and frequency domain. The files contained in this example can be downloaded from the Ansol technical support website in the SAMPLESRearAxleDynamics directory. The session file, RearAxleRollerBearing_dynamic_feprobes.ses, requires the rearaxle_housing and carrier_constraints.bdf mesh files be located in the working directory.

### 18.1 Setting up the FEProbes

The FEPROBES menu, shown in Figure ??, is accessed from the main T3D Guide menu, and is used to define the probe location and data component to be written to the probe file. The filename to which all probe data is written is defined within the FEPROBE menu, and must be identical for all probes. The probe location is defined by the BODY, MESH, INSTANCE of the mesh, ELEMENT within the instance, and element coordinates XI, ETA, and ZETA. These inputs are obtainable from a pre-processing iGlass file by double clicking on the desired node and scrolling down in the ‘Attributes’ tabbed window. Each probe can be used to output one data component. In this example we would like to write the 3 displacement components, 3 velocity components, and 3 acceleration components at 2 probe locations. A total of 18 probes are used, with the FEPROBES.DAT output file containing a column of time values, followed by columns of component values for each probe at each time instance. The total size of the array in the output file is NTIMSTEPS rows X (NFEPROBES + 1) columns.

### 18.2 Plotting FEProbe Time and Frequency Response

A Matlab script for visualizing the probe output is included in the SAMPLESRearAxleDynamics directory on the tech support website. The script is written to read an FEPROBE output file named FEPROBES.dat. The script generates a plot containing the time and frequency response for each probe and writes it to a file within a folder called DynamicResponsePlots. The user selects the number of samples from the end of the data to use for the FFT calculation, N. A sample of the plot generated is shown in Figure 18.2.
Figure 18.1: The FEPROBES menu.
Figure 18.2: Time and frequency response of output probe data.
In this chapter, we look at how to process time domain response data from a Transmission3D dynamic analysis to detect the presence of bearing defects. The model files used in this example can be found in the /SAMPLES/SimpleBearingDynamics folder in the T3D repository on the tech support website. Based on published literature by Lewicki [6] and McFadden [7], the enveloping technique is useful to detect the presence of bearing defects, by analyzing the vibration data from a stationary housing mounted accelerometer. It helps to separate the bearing fault signature that is masked by other stronger vibrations in the response.

The steps involved in the enveloping technique are shown in the Figure 19.1. The raw signal from the accelerometer is passed through a band pass filter to retain the transient system response caused by the impulse excitation that occurs when a roller goes over a pit (and remove other frequencies). This impulse excitation occurs periodically during bearing operation. The frequency range of the band pass filter must be chosen carefully to include the resonance frequency of the dominant mode of vibration.

The filtered signal is then demodulated using an envelope detector and converted to frequency domain. The presence of a bearing characteristic frequency in the FFT can be associated with the development of a defect on the inner race, the outer race or the roller.
To demonstrate the signal processing procedure, the above steps are carried out on a simulated signal, which is generated as a sum of the impulse response signal, sinusoidal system vibration and noise. The impulse response signal is a damped sinusoid that occurs whenever the pit comes into contact in the load zone. The system vibration represents the steady state vibration (in the absence of bearing fault) and is generated by a combination of sine waves at three different frequencies and phase shifts. Finally, random noise is added to the other signals to generate the simulated system response.

As illustrated in Figure 19.1, the simulated raw signal is passed through a band pass filter to extract the transients created by the impulse. The selection of band pass filter frequency range is critical to extracting the impulse response from raw data. In a real transmission system, this is expected to be closer to one of the resonance frequencies of the system.

The filtered signal is then passed through the envelope detector. We tried two methods for envelope detection. The first method uses a Hilbert transform (shown in Figure 19.1 and Figure 19.2a) while the second method uses a full wave rectification followed by a low pass filter (Figure 19.2b). Both methods for envelope extraction work well. The Hilbert transform performs better in tracing the amplitude of a band pass signal compared to the low pass filter method, as shown in Figure 19.2.

The frequency spectrum of the resulting envelope signal from both methods are similar and shows spectral content at impulse repetition frequency. (second plot in Figure 19.2a and Figure 19.2b).

### 19.1 Model Setup

For this example, we started with a simple system to understand how the model parameters must be tuned to get a proper response of the physical system that does not contain any modeling artifacts. A Transmission3D model of a
Figure 19.2: Two methods evaluated for envelope detection.

For this configuration, the following are the rotational speeds and time periods associated the inner race, outer race, the cage and the individual roller.
Figure 19.3: Simple system with one roller bearing.
Table 19.1: Simple bearing model - System information.

<table>
<thead>
<tr>
<th>Inputs</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Inner Race Outer Radius, mm</td>
<td>20.5</td>
</tr>
<tr>
<td>Outer Race Inner Radius, mm</td>
<td>29.5</td>
</tr>
<tr>
<td>Roller Diameter, mm</td>
<td>9</td>
</tr>
<tr>
<td>Number of Rollers</td>
<td>17</td>
</tr>
<tr>
<td>Cage Radius, mm</td>
<td>25</td>
</tr>
<tr>
<td>Inner Race RPM</td>
<td>600</td>
</tr>
<tr>
<td>Outer Race RPM</td>
<td>0</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Derived Quantities</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Inner Race - Time for one rotation, sec</td>
<td>0.1</td>
</tr>
<tr>
<td>Cage RPM</td>
<td>246</td>
</tr>
<tr>
<td>Cage - Time for one rotation, sec</td>
<td>0.2439</td>
</tr>
<tr>
<td>Roller Pass Period / Outer Race, sec</td>
<td>0.01435</td>
</tr>
<tr>
<td>Roller Pass Frequency / Outer Race, Hz</td>
<td>69.7</td>
</tr>
<tr>
<td>Roller Pass Period / Inner Race, sec</td>
<td>0.00997</td>
</tr>
<tr>
<td>Roller Pass Frequency / Inner Race, Hz</td>
<td>100.3</td>
</tr>
</tbody>
</table>

The results of roller load distribution from a one step static analysis are shown in the Figure 19.4. It is seen that there are seven rollers in the load zone which extends from $\theta = -74^\circ$ to $\theta = +74^\circ$.

![Roller bearing load distribution](image1)

(a) Loads carried by the individual rollers.  
(b) Load distribution function.

Figure 19.4: Roller bearing load distribution from static analysis

19.1.1 Measured Signal

In prior studies, the **rigid body deflection** of the inner race relative to the outer race was analyzed of bearing defects. This measurement was not capturing the local effects due to the change in loading when the pit enters a load cycle.

For this simple bearing model example, instead of using bearing deflection, **finite element (FE) probes** were instrumented at multiple locations around the bearing outer race (on the outer diameter).
The FE probe measures the deformation at the probe location in the UX, UY, UZ and UR directions. The deformation signal measured by the FE probe is similar to the signal measured by a stationary accelerometer mounted on the housing. The FE probes were mounted close to the contact region (on the outer diameter of the outer race), so they can capture the signal before it is attenuated as we move farther away from the contact zone. In this example four probes were placed 45 degrees apart (Figure 19.5) around the bearing load zone. From previous studies, it is known that the measurement location is important in getting a quality signal from the system (Lewicki [6]).

19.1.2 Race Modeling

The next change to the model was to refine the finite element modeling near the FE probes to improve the response. The FE probe is placed on the outside of an outer race where it connects with supporting rotor. A Fourier expansion is used to connect the two different meshes at this point. If the race has a low order Fourier expansion, it acts as a spatial filter that artificially stiffens the race and cuts off the high frequency modes that are not in the basis. To avoid this, and improve the deflection at this interface, circular and axial orders of the race were increased.

Corresponding refinements need to be made in the finite element resolution of the race (number of elements in the circular and axial directions). For this model the number of circular divisions on the outer race is increased to 128 and circular Fourier order of 68 was chosen. The number of elements on connecting rotor were also increased to model the high Fourier order of the race.

After refining the finite elements and improving the signal, we observed that the steady state probe response was sinusoidal instead of a flat line (that was obtained with an earlier low Fourier order race model). However, we observed small spikes in the response as the contact rolls over the finite element boundaries. This is a numerical artifact introduced by the finite element representation of the outer race, and cannot be avoided. It can be seen that increasing circular divisions around the race can reduce the amplitude of these spurious spikes. This effect is shown in Figure 19.6 where the probe response with 128 and 512 circular divisions is shown. Increasing the number of divisions will contribute to small increase in run time. Based on the model and requirements, appropriate refinement parameters can be chosen.
We also checked to make sure if the finite element size in the radial direction is small enough to allow the pressure waves to go through. The velocity of P-waves in a homogeneous isotropic medium is given by

\[ v_p = \sqrt{\frac{\lambda + 2\mu}{\rho}} \]  

(19.1)

The velocity of P-wave in iron is approximately 5200 m/s. At a frequency of 50 kHz, the wavelength of the P-wave is 10.4 cm. Assuming that we need twelve linear elements per wavelength to capture it accurately, we are fine as long as the radial finite element length is less than 8.5 mm. At a radial element length of 1.3 mm, our mesh is sufficiently fine to resolve P-waves.

19.1.3 Time Step

The last change is to choose the time step required for the dynamic analysis. Based on the enveloping technique, it is understood the impulse is carried by one of the higher order system resonances. In order to capture this effect, time step must be small. This will ensure the impulses from bearing faults are seen in the time domain signal. For this model, a time step of 20 microseconds is chosen. This sampling interval corresponds to sampling rate of 50 kHz that was used in experiments by Lewicki [6].

19.2 Bearing State Tool

After performing the Transmission3D analysis, and looking at the time response, and trying to make sense of it, it is very useful to have a simple tool that shows the spatial location of the inner race, rollers and outer race at any time instant. This helps in correlating observed changes in the response back to the physical events happening at a particular instant of time.
We have developed a light weight Matlab tool for this purpose. It takes simple geometry (OD of inner race, ID of outer race, number of rollers) and speed (angular speed of the inner and outer races) inputs along with the defect information. Any any time instant, we can plot the state of the bearing as shown in Figure 19.7.

![Figure 19.7: Screenshot of bearing state tool.](image)

19.3 Baseline model

With the modeling updates described above, the simple bearing model was run without any defects to establish a baseline. Figure 19.9 shows the response after initial transients have died down and its frequency spectrum. The component shown is the UX deformation at Probe 1.

The major frequency of the signal is equal to the outer roller pass frequency (69.7 Hz). This is expected since the probe is on a fixed outer race and signal repeats as each roller passes over it. It is observed the UX is maximum when the roller is directly under the probe and reaches minimum when the probe is in between two rollers. For this system without an other excitations, the frequency spectrum shows just the first harmonic of the outer roller pass frequency. This can be used a reference to compare the response from bearing models with defects.

The modeling improvements have shown a significant improvement in the measured response signal, as shown in Figures Figure 19.9 and Figure 19.8. This gives us confidence to simulate the cases when defects are included.
In this section a pit is applied to outer race, inner race and roller of single bearing model to demonstrate the extraction of fault signal using enveloping technique from the output dynamic response. The above discussed single bearing model is used for all the three cases. The pit is on the X axis at the $\theta = 0$ for all three cases as shown in Table 19.2. The pit is shown as a notch and black lines represent the approximate load zone of the bearing due to the pure axial
Table 19.2: Bearing pit conditions

<table>
<thead>
<tr>
<th>Case</th>
<th>Pit Location at $t = 0$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Outer Race Defect</td>
<td><img src="outer_race_defect.png" alt="Diagram" /></td>
</tr>
<tr>
<td>Inner Race Defect</td>
<td><img src="inner_race_defect.png" alt="Diagram" /></td>
</tr>
<tr>
<td>Roller Defect</td>
<td><img src="roller_defect.png" alt="Diagram" /></td>
</tr>
</tbody>
</table>

load in the X direction. The dimension of the elliptical pit are 4mm wide along the roller length, 1 mm wide in circular direction and 0.5mm deep. The X deflection from probe 1 is referred to as response in all the cases for this section.

The UX response from probe 1 are shown in Figure 19.10. For the outer race defect, the pit on a stationary outer race is always under the probe 1. As each roller goes over the pit, it can be clearly seen in the time domain signal after initial transient region. The vertical blue lines are spaced by time between each roller passing a same point on the outer race. In case of inner race defect, the pit is on a rotating member and the fault signal with the maximum amplitude occurs when the pit aligns with probe in X axis. In this model, inner race is rotating at 600 rpm which corresponds to 0.1 sec for one full rotation. After each 0.1 sec the pit comes back to align with the X axis. The defect at this time instant can be seen in the time domain graph without any data processing. For the roller defect, the time for the cage to go one full rotation is 0.2439 sec. This is when the peak amplitude is picked up by probe 1. Both for inner race defect and roller defect, the pit hit several other times in the load zone and cannot be noticed from the pure time domain signal. The enveloping technique comes useful to extracting defect signals under these conditions from raw time signal from probes.

First we will demonstrate the enveloping technique on outer race defect. (todo: add figure 158 shows the transient spectrun without enveloping) The signal is band pass filtered at center frequency of 1500 Hz with bandwidth of 1kHz. This extracts impulse signal from the full system response. This signal is then pass through the envelope detector using Hilbert transform and spectrum analyzer. The spectral plot is shown in Figure 19.11. From the frequency that appears in the spectral plot, we confirm the location of the defect is on the outer race.

Figure 19.14 shows the frequency spectrum of the raw signal without any data processing. For the inner race defect, the enveloping technique is shown in Figure 19.13. For the inner race defect, pit hits the roller at three times in the load zone for one full rotation. The instantaneous position of pits where it contacts roller in the load zone is show in Figure 19.15. The envelope curve shows the three distinct hits for each full rotation of the inner race. Since the hit 2
Figure 19.10: Dynamic response from the models with bearing pits

is closest to the probe 1, the amplitude of this signal is higher than the other two hits. The amplitude is also dependent on load carried by the roller. The frequency spectrum of the envelope curve shows peaks at inner race shaft frequency and at first harmonic of inner race ball pass frequency.

todo. Figure 163 shows The roller defect data processing is demonstrated in Figure 19.16. The pit is modeled on roller 1 only. The roller pit passes over the inner and outer race combined 5 times. This was seen from the UX response at probe 1. Here the spectral lines shows up at the cage frequency and twice ball spin frequency.
Figure 19.11: Envelope extraction from a signal with outer race defect

Figure 19.12: Frequency Spectrum of outer race defect signal without data processing
Figure 19.13: Envelope extraction from a signal with inner race defect

Figure 19.14: Frequency Spectrum of inner race defect signal without data processing
Figure 19.15: Location of inner race pits in the load zone contacting rollers
Figure 19.16: Envelope extraction from a signal with roller defect

Figure 19.17: Frequency Spectrum of roller defect signal without data processing
A concentrated load on a shaft segment is often useful in order to simply model shaft loading while reducing the model size and computational resources associated with detailed contact bearings and/or gears. This example demonstrates the use of shaft segment point loads for these scenarios.

### 20.1 Defining a Point Load on a Shaft Segment

A point load is created by selecting the ENABLEPOINTLOAD boolean box inside of the shaft SEGMENT menu of the segment where the load is desired (Figure 20.1). After selecting the box, the POINTLOAD submenu will appear. The POINTLOAD menu, shown in Figure 20.2 is used to define the location, orientation, method, and magnitude of the load relative to the shaft segment origin. For this example, we define a load in the center of the 55 mm long shaft segment by setting \( ZPOS = 27.5 \), and define the load on the outer surface of the shaft segment pointing in the direction of the negative X-axis \( (XPOS = 27.5, AX = -1) \). The OUTSIDECONTACT box must also be selected in the SEGMENT menu to enable the point load at the outer surface.

Verification that the point load is properly modeled can be obtained from a post-processing iGlass file by turning on the stress distribution. A local stress at the point load location should appear as shown in Figure 20.3. Figures 20.4 and 20.4 show the effect of the point load on the gear tooth contact pattern.

### 20.2 Using a Point Load as a Sensor

A point load can also be used as a displacement probe at a particular location of interest. Enabling the point load as described above, but setting the load value to zero, allows a user to obtain the displacement at the point load location from the CONCLOADS.DAT file written to the calyxtmp/ folder during the model analysis stage. A sample of the file output is provided in Figure 20.6.
Figure 20.1: The shaft segment menu with point load enabled.

<table>
<thead>
<tr>
<th>SEGMENT</th>
<th>INNERDIA</th>
</tr>
</thead>
<tbody>
<tr>
<td>3</td>
<td>10.0000000000000000000</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>TYPESEG</th>
<th>DEFINEGEOMETRY</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>LENGTH</th>
<th>OUTERSHAPE</th>
</tr>
</thead>
<tbody>
<tr>
<td>55.0000000000000000000</td>
<td>CYLINDRICAL</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>OUTERDIA</th>
<th>EDGERAD1</th>
</tr>
</thead>
<tbody>
<tr>
<td>55.0000000000000000000</td>
<td>0.0000000000000000000</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>EDGERAD2</th>
<th>OUTERCURVATURE</th>
<th>INNERSHAPE</th>
<th>INNERDIA</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.0000000000000000000</td>
<td>0.0000000000000000000</td>
<td>CYLINDRICAL</td>
<td>10.0000000000000000000</td>
</tr>
</tbody>
</table>

Figure 20.2: The shaft segment point load menu.

<table>
<thead>
<tr>
<th>POINTLOAD</th>
<th>XPOS</th>
<th>YPOS</th>
<th>ZPOS</th>
<th>AX</th>
<th>AY</th>
<th>AZ</th>
<th>POINTLODMETHOD</th>
<th>LOADMAGNITUDE</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>22.5000000000000000000</td>
<td>0.0000000000000000000</td>
<td>22.5000000000000000000</td>
<td>-1.0000000000000000000</td>
<td>9.0000000000000000000</td>
<td>9.0000000000000000000</td>
<td>INTERSECTION</td>
<td>10.0000000000000000000</td>
</tr>
</tbody>
</table>
Figure 20.3: The local stress concentration at the point load location.

Figure 20.4: The contact pattern before applying the point load.
Figure 20.5: The contact pattern after applying the point load.

Figure 20.6: The CONCLOADS.DAT output file with displacement values at the point load location.
The condensed housing option allows a user to reduce a finite element housing structure to a stiffness matrix containing only the desired degrees of freedom at a set of master nodes. In the condensed housing example, we condensed a finite element housing model down to a stiffness matrix containing 6 master nodes, each with 6 degrees of freedom. In this example, we demonstrate how to specify a load/displacement boundary condition at any master node. This feature is particularly useful when a user would like to model housing thermal expansion with a condensed housing stiffness matrix.

### 21.1 Defining a Displacement Boundary Condition at a Master Node

A displacement boundary condition can be specified at each master node of the condensed housing by selecting the UNLOADEDDEFM boolean box as shown in Figure 21.1. Selecting the box displays an additional UNLOADED-DEFM_SCALEFACTOR that allows for universal scaling of the displacement values. The displacement values are entered within the NODE menu for each node as shown in Figure 21.2. Figure 21.3 shows the effect of the displacement on the bearing stress distribution.
Figure 21.1: The condensed housing menu with retained node displacement enabled.

Figure 21.2: The condensed housing node menu with displacement.
Figure 21.3: Undeformed housing model (left) vs nodal displacement model (right) results.
21.2 Defining a Load Boundary Condition at a Master Node

A load at a master node is applied in a similar manner to a displacement. Selecting the LOAD boolean box, as shown in Figure 21.4, allows for the specification of force and moment components within the NODE menu (Figure 21.5). Figure 21.6 shows the effects of applying nodal forces at the housing master nodes.
Figure 21.5: The condensed housing node menu with force.

Figure 21.6: Unloaded housing model (left) vs nodal load model (right) results.
In this example, we demonstrate the max damage criterion in the fatigue postprocessing menu. A detailed description of this criterion is provide in the Mulyx Pre and Post-processing Manual. For this example, we modified the hypoid gear rim in the Rear Axle example from this document to include a press-fit connection between the base of the rim and a cylindrical surface on the carrier. An interference is then specified to introduce a positive stress at the hypoid gear tooth fillet. A schematic of the updated rim model with press-fit is shown in Figure 22.1.

22.1 Max Damage Fatigue Post-processing

The max damage criterion is selected in the FATIGUE post-processing menu by selecting MAX DAMAGE from the CRITERION drop-down menu. Figure 22.2 shows the fatigue menu inputs for the max damage criterion. We select the hypoid gear fillet as the surface of interest, and enter a DISTMIN of $0.25 \times \text{ToothHeight}$. Figures 22.3 and ?? show the critical point stress for the original rim model without press-fit and the updated model with press-fit. The model with press-fit shows a vertical shift due to the increase in nominal stress in the fillet region the results from the press-fit interface.
Figure 22.1: The modified rear axle model hypoid gear rim with press-fit at carrier.

Figure 22.2: The FATIGUE post-processing menu with MAX_DAMAGE criterion selected.
Figure 22.3: Critical point stress plot without press-fit.

Figure 22.4: Critical point stress plot with press-fit.
REFERENCES


