# Table of Contents

- Bolt Toolkit toolbar ........................................................................................................................................... 3
- Bolt Toolkit Help .................................................................................................................................................. 3
- News in V180.8 .................................................................................................................................................... 3
- Rivet Group ......................................................................................................................................................... 4
- Rivet Group Result ............................................................................................................................................. 8
- Bolt Group Pretension ....................................................................................................................................... 10
- Simplified Bolt Group ....................................................................................................................................... 12
- Advanced Bolt Group ........................................................................................................................................ 16
- Bolt Group Result .............................................................................................................................................. 22
- Mapping of thermal loads ................................................................................................................................. 26
- Resolve solution convergence problems for simplified bolts and rivets ......................................................... 27
- References ....................................................................................................................................................... 28
**Bolt Toolkit toolbar**

The main toolbar has six action buttons, **Rivet Group**, **Rivet Group Results**, **Bolt Group Pretension**, **Simplified Bolt Group**, **Advanced Bolt Group**, **Bolt Group Results** and **Bolt Toolkit Help** that are described in the following sections.

![Bolt Toolkit](image)

**Bolt Toolkit Help**

This button opens the help documentation (this file).

Technical Enhancements and Customer Support, TECS, is not included in the license. However, the user may use this link to report any bugs or enhancement requests ansys@edrmesodo.com and make sure to include “Bolt Toolkit” in the subject line.

A demo model, “BoltToolkit_V180.8.wbpz”, is included in the extension zip file.

For information about element types used in the tools or other FEM details use the ANSYS online help [1].

**News in V180.8**

- Bugfix for bolt results at head and thread for cyclic symmetry.
- Bolt thread node position factor for simplified bolts.
  0: Beginning of thread, 0.5 Mid of thread (default), 1: End of thread
- Improved bolt hole centroid calculation for multi-edge holes.
- Improved bolt position using thread axis or nut plane to position head in oversized or slotted holes.
**Rivet Group**

**Background**
Riveted assemblies are hard to model in Mechanical since you need to connect every edge pair along the rivet with e.g. a fixed body to body joint.

**Solution**
The Rivets Group introduces a new load feature where so called “Rivets” are created and placed in a group folder in the current static analysis. The Rivet is constructed of “BEAM188” solid circular for the rivet shaft and a “CERIG” or “RBE3” for the heads and intermediate sections. This is the similar as the standard “Beam connection” except that rivet creates one beam element for each layer in the connected assembly.

The additional features with the “Rivets Group” compared to “Beam connection” are that:

- Multiple rivets can be defined in one load object, i.e. all M12 rivets of the model
- A point mass can be added to the rivet head to account for missing mass “MASS21” element
- All rivets are saved in one place, the “Rivets Group” folder
- The rivets can have different orientation
- Custom “Behavior” for the “head” to “Shaft” interaction can be defined in an APDL macro.
**Usage**

Clicking on the Rivets Group in the toolbar inserts the “Rivets Group” folder in the current static analysis as well as creating the first Rivets load object.

To add additional rivets, right click on the Rivets Group folder and select Add Rivets.

You can also use “Duplicate” on an existing Rivets and modify the properties.

Inputs to define the Simplified Bolts are:

- **Rivets Holes**: Circular edges connected to non-cylindrical faces. It can also be one Named Selection. Both shell and solid bodies are supported.
- **Material**: Select material name from the list of available materials from Engineering Data.
- **Head Mass**: Add the missing mass (optional). Default calculated to the mass of a sphere section based on steel density. The sphere section diameter, D, is the “Head Diameter” and the sphere section height, h, is D/(2√3), see the figure above.
- **Shaft Diameter**: Rivet diameter that defines the circular solid section for the beam element. By default the minimum diameter from the hole selection is filled in.
- **Head Diameter**: Sets the maximum size of the rivet heads. By default two times the shaft diameter is used.
- **Max thickness**: Search distance for creating rivets, i.e. maximum part thickness along the rivet shaft.
- **Behavior**: Rigid/Deformable/Custom (*).

**Notes on Material**: The following material properties (and corresponding MAPDL Lab) are supported:

- Young’s Modulus (EX), Poisson’s Ratio (NUXY), Density (DENS), Coefficient of Thermal Expansion (ALPX), Thermal Conductivity, (KXX), Specific Heat (C), Isotropic Hardening (BISO & MISO), Kinematic Hardening (BKin, KINH)

All inputs are checked to be valid to avoid error during solution.

At least two sets of hole edges are needed to create a valid selection.

(*) Custom behavior can be defined in the MAPDL macro file “buildRivet.mac”, see following pages.
Graphics
A graphic representation is created once valid inputs are given. The **Rivets Group** load object searches to connect the holes with a distance less than *Maximum thickness*. Hole centroids with a distance less than 1% of the “Maximum thickness” are made coincident. The active hole edges are plotted in thick red lines. The head diameter is plotted as a green disc (normal to the rivet shaft) and the rivet shaft as a thick green line. Un connected hole edges are plotted in thin red lines.
**Custom behavior**

The rivet connection behavior is defined in the macro file “buildRivet.mac” found in the extension installation folder. (%USERPROFILE%\AppData\Roaming\Ansys\v180\ACT\extensions\BoltToolkit_V180.8)

Edit the file in the sections “*else ! 'Custom’*” to define any type of connections using the center node on the rivet shaft (_npilot) and the selected edge nodes.

![BuildRivet macro file](image)

**Known Issues/limitations**

Drag and drop of a Rivets Group object or individual rivets between analyses is not supported. You may duplicate an analysis system and you may duplicate an individual rivet inside a Rivets Group.

The Rivets Group can be used in a pre-stressed **Modal** analysis by using Rivets in the **Static Structural** analysis. It is not enough to link the model cell from the static analysis to the **Modal** analysis. You must also link the **Solution** cell from the **Static** analysis to the **Setup** cell of the **Modal** analysis since the bolt is treated as a “load object”.

A downstream modal superposition analysis, e.g. **Harmonic** analysis, cannot use surface based loads applied by “Surface Effect”, use “Direct” instead. (“Direct FE” and “Remote point” loads works).
**Rivet Group Result**

The **Rivet Group Result** object adds the post processing features to the **Rivet Group** load object. This result object will extract all forces and moments in all rivet sections for a selected group of rivets and plot selected result on the model and create a results summary listing.

**Usage**

Inputs to the **Rivet Group Result** are:

- **Geometry**: All Bodies (default) or selection by scooping or named selection (*).
- **Rivet Group**: A list box of all active rivet group objects. To avoid duplicate names the rivet objects Id number is appended to the name. When selecting a rivet group the corresponding geometry is automatically selected.
- **Rivet Result**: A list box of results to display (Default “Normal Force”)
  - Normal Force/Shear Force/Normal Stress/Shear Stress
- **By**: Time/Result Set/Maximum Over Time/Time Of Maximum (standard Mechanical feature)
  - Note that you may use Maximum Over Time to get the worst case from all steps. In addition, the Graph displays the time history of min and max for the selected result.
- **Display Time**: Load step time for results evaluation. You may only use time steps where there are results saved, see the graph and tabular data window for load steps to use.

(*) The node numbers from the selected rivet group will be compared with the node numbers in the selection and results will only be plotted for nodes/rivets that are scooped in the selection.

Note that “Nodal Forces” must be set to “Yes” in **Analysis Settings** under **Output Controls** in order for post processing to work. When adding the first rivet this setting is automatically set to “Yes”.
Output
The selected “Rivet result” is displayed on each edge section for all rivets.

The Rivet Group Result object also writes a result listing to the solver files directory of the analysis. The naming of the output file is based on the selected Rivet Group Result name and an additional internal unique id number in case duplicate names, e.g. “Rivet_Group_Result_Id4553.lis”. This listing includes information about all results sets, the rivet shaft section node, forces and moments and the resulting stresses.

The rivet forces are extracted at all sections along the rivet. At coincident sections only one side is included in the listing.

In the output listing there is one line for each rivet section that is identified by the internal shaft section node number and the location of the section in global coordinates.

The header shows the name of the analysis system (Static Structural) and the name of the rivet group object (Rivets_Id4553). The columns of the file contains the following information:

| Set: | Result set number. See the Tabular data window for Set number and corresponding Time In
dex: | Rivet section index. |
| Node id: | Shaft node of the rivet |
| X, Y, Z: | Global coordinates of the rivet section |
| FX shear: | Local shear force in X direction |
| FY shear: | Local shear force in Y direction |
| FZ normal: | Normal force in the bolt section |
| MX transv: | Local translational (bending) moment around X axis |
| MY transv: | Local translational (bending) moment around Y axis |
| MZ axial: | Axial torsion moment in the bolt section excluding nominal pretension moment |
| S normal: | Normal stress in rivet section, \( S_{\text{normal}} = \frac{F_{\text{normal}}}{A} = \pi \text{Diam}^2/4 \) |
| S shear: | Shear stress in bolt section, \( S_{\text{shear}} = \frac{F_{\text{shear}}}{A} \) |

Notes on Output
To speed up post processing of the rivets results and properties are exported during solution of the model. In the solution folder you will find a rivet data text file, e.g. “Rivets_Id4553.rivet” that contains FE-model data for the selected rivet group and a results listing, e.g. “rivetRes_Id4553.lis”, in solution units.

The file “rivetdata_header.csv” contains the header line with information about the columns in the rivet data file; # Rivet node,X,Y,Z,Beam elem,Head elem” and the file “rivetRes_header.csv” contains the header line for the results listing. # Nset, Index, Rivet node, X, Y, Z, FX, FY, FZ, MX, MY, MZ”.

- 9 -
**Bolt Group Pretension**

**Background**

Big assemblies with many bolt joints takes a lot of time to prepare as you have to manage either solid bolts that has to be meshed and connected with contacts or the Mechanical feature “Beam connection”. In addition, bolt pretension will result in many individual load objects for the solid bolts. A “Beam connection” requires specialist APDL knowledge in order to apply pretension using “user command objects”, UCO.

One additional drawback with the pretension load in Mechanical is that it does not use the “SLOAD” feature to save the pretension load history in the model but applies the load as force during the “apply” step and lock it using displacement at the “lock” step. This information is not saved if you archive the model using the “CDWRITE” command. Note that all other loads and boundary conditions are applied using tabular loads, i.e. the load history is saved with the model.

**Solution**

The “Bolt Group Pretension” load object can be used to simultaneously apply the same bolt pretension to multiple solid or line body bolts. The pretension load is defined using “PRETS179” elements and section loading “SLOAD”. The bolts can have different orientation and the pretension normal will be set for each individual bolt. A visual confirmation is drawn as a red arrow for each bolt to show the pretension normal.

Post processing of the “Bolt Group Pretension” can be done using the “Bolt Group Results” object, even if the pretension value is set to zero.
Usage
Inputs to define the Bolt Group Pretension are:

- **Bolt shaft**: One or many cylindrical solid body faces OR one or many straight line body edges. It can also be one Named Selection.

- **Pretension Force**: Force value in the “SLOAD” command. If zero, the pretension section is locked.

- **Load Step Apply**: Load step to apply the force. Used only if *Pretension* > 0.

- **Load Step Lock**: Load step to lock the force to a displacement. Used only if *Pretension* > 0.

All inputs are checked to be valid to avoid error during solution, i.e. the analysis must have at least two load steps to use the pretension force. You can use the pretension force and only one load step if you set “Load Step Apply” = 1 and “Load Step Lock” = 2. In this case the bolt will not be locked as only the force is applied in the analysis.

**Known Issues/limitations**
Geometry cannot be a “Virtual Topology” item.

The pretension can only use the “Force and Lock” feature of the “SLOAD” command, not the “Displacement” or “Increment” feature.

The Bolt Group Pretension can be used in a pre-stressed Modal analysis by using Bolt Group Pretension in the Static Structural analysis. It is not enough to link the Model cell from the static analysis to the Modal analysis. You must also link the Solution cell from the Static analysis to the Setup cell of the Modal analysis since the bolt is treated as a “load object”.

A downstream modal superposition analysis, e.g. Harmonic analysis, cannot use surface based loads applied by “Surface Effect”, use “Direct” instead. (“Direct FE” and “Remote point” loads works).
**Simplified Bolt Group**

**Background**

Big assemblies with many bolt joints takes a lot of time to prepare as you have to prepare mesh, contacts and pretension for every bolt. Solid bolt geometry may not be suitable for meshing and require geometry clean-up to fix diameters and remove small features. It will also add many degrees of freedom to the model.

**Solution**

The Simplified Bolt Group introduces a new load feature where so called “Simplified bolts” are created and placed in a group folder in the current static analysis. The Simplified bolt is constructed of two “BEAM188” solid circular for the bolt shaft and a “CERIG” or “RBE3” (same rigid remote point) for the head and nut/thread. This is the same as the standard “Beam connection” except that beam connection uses remote point to connect to the beam.

The additional features with the “Simplified Bolt Group” compared to “Beam connection” are that:

- Multiple bolts can be defined in one load object, i.e. all M12 bolts of the model
- A point mass can be added to the bolt head to account for missing mass “MASS21” element
- Pretension load can be applied and is defined using “PRETS179” elements and section load “SLOAD”
- All Simplified bolts are saved in one place, the “Simplified Bolts Group” folder
- The bolts can have different orientation and the pretension normal will be set for each individual bolt
- Custom “Behavior” for the “head” to “thread” interaction can be defined in an APDL macro.
- Thermal loads on the bodies will be mapped to the bolts to account for correct elongation.
- Post processing of the simplified bolts can be done using the “Bolt Group Results” object
**Usage**

Clicking on the **Simplified Bolts Group** in the toolbar inserts the “Simplified Bolts Group” folder in the current static analysis as well as creating the first **Simplified Bolt** load object.

To add additional bolts, right click on the **Simplified Bolts Group** folder and select **Add Simplified Bolts**. You can also use “Duplicate” on an existing Simplified Bolt and modify the properties.

![Image of Simplified Bolts Group in toolbar]

Inputs to define the Simplified Bolts are:

- **Bolt Head**: Planar circular or conical faces OR edges connected to non-cylindrical faces. It can also be one Named Selection. Both shell and solid bodies are supported.
- **Nut/Thread**: Planar circular or conical faces OR edges connected to non-cylindrical faces (Nut) OR cylindrical faces (Thread). It can also be one Named Selection. Both shell and solid bodies are supported.
- **Material**: Select material name from the list of available materials from Engineering Data.
- **Head Mass**: Add the missing mass (optional).
- **Head Diameter**: Sets the maximum size of the bolt head and nut in case of edge selection. If using a surface selection, make sure that the diameter is equal or larger than the face diameter.
- **Shaft Diameter**: Bolt shaft diameter that defines the circular solid section for the beam element.
- **Behavior**: Rigid/Deformable/Custom (*)
- **Thread Node Position**: Thread node relative position 0: Beginning, 0.5: Middle (default) 1: End
- **Pretension Force**: Force value in the “SLOAD” command. If zero, the pretension section is locked.
- **Load Step Apply**: Load step to apply the force. Used only if Pretension > 0.
- **Load Step Lock**: Load step to lock the force to a displacement. Used only if Pretension > 0.

**Notes on Material**: The following material properties (and corresponding MAPDL Lab) are supported: Young’s Modulus (EX), Poisson’s Ratio (NUXY), Density (DENS), Coefficient of Thermal Expansion (ALPX), Thermal Conductivity, (KXX), Specific Heat (C), Isotropic Hardening (BISO & MISO), Kinematic Hardening (BKin, Kinh)

(*) Custom behavior can be defined in the MAPDL macro file “simpleBolt.mac”, see following pages.

All inputs are checked to be valid to avoid error during solution, i.e. the analysis must have at least two load steps to use the pretension force. If the force is zero, no pretension section is created and one load step is enough. You can use the pretension force and only one load step if you set “Load Step Apply” = 1 and “Load Step Lock” = 2. In this case the bolt will not be locked as only the force is applied in the analysis.
The head and nut/thread must have equal number of selected entities and also one unique match between head and nut. It is not allowed to have two bolt heads connecting to the same thread. This will be shown as invalid thread selection.

**Graphics**

A graphic representation is created once valid inputs are given. The Simplified Bolt load object searches to connect the head and nut with the minimum distance. The head geometry is plotted in red and the nut/thread in blue. The head and nut diameter is plotted as a green disc (normal to the bolt shaft) and the bolt shaft as a green line.

Note: The head centroid (and disc) is adjusted to be on the axis from the corresponding thread/nut. If selecting non-matching head and thread geometry the app may still create valid bolts if the bolt head plane is the same.
Custom behavior

The bolt connection behavior is defined in the macro file “simpleBolt.mac” found in the extension installation folder. (%USERPROFILE%\AppData\Roaming\Ansys\v180\ACT\extensions\BoltToolkit_V180.8)

Edit the file in the sections “*else ! ‘Custom’” to define any type of connections using the center node on the bolt shaft (_npilot) and the selected face nodes.

Known Issues/limitations

The pretension can only use the Force and Lock feature of the SLOAD command and not the Displacement.

Drag and drop of a Simplified Bolt Group object or individual bolts between analyses is not supported. You may duplicate an analysis system and you may duplicate an individual bolt inside a Simplified Bolt Group.

The Simplified Bolt Group can be used in a pre-stressed Modal analysis by using Simplified Bolt in the Static Structural analysis. It is not enough to link the model cell from the static analysis to the Modal analysis. You must also link the Solution cell from the Static analysis to the Setup cell of the Modal analysis since the bolt is treated as a “load object”.

A downstream modal superposition analysis, e.g. Harmonic analysis, cannot use surface based loads applied by “Surface Effect”, use “Direct” instead. (“Direct FE” and “Remote point” loads work).
Advanced Bolt Group

Background
The Simplified Bolt Group is good to connect parts and apply pretension but it is not suitable to study the stress locally around the bolts since it uses rigid connections. To do detailed studies of bolt joints a solid model of the bolt is needed with appropriate meshing, contact setup and pretension load. For a big assembly this will take a lot of time to prepare as you have to do it for each bolt.

Solution
The Advanced Bolt Group introduces a new load feature where so called “Advanced Bolts” are created and placed in a group folder in the current static analysis. The Advanced Bolts are constructed of “SOLID185” brick elements with frictional contact (or bonded) at the head and the new bolt thread contact at the thread. The thread contact section introduces the “wedge” effect when the bolt is loaded. The geometry is based on ISO-15071 “Hexagon bolt with flange” or ISO-4762 “hex socket head cap screw”. It is also possible to create a bolt/nut configuration.

Custom bolt dimensions can easily be added by creating a new csv-file, see section “Custom Bolt Dimensions”.

In addition to how the Simplified Bolts works, the advanced bolt dimension is based on a list of valid bolts and the pretension load can be calculated from the specified friction coefficient and pretension torque.
Usage
Clicking on the Advanced Bolts Group in the toolbar inserts the “Advanced Bolts Group” folder in the current static analysis and creates the first Advanced Bolts load object.

To add additional bolts right click on the Advanced Bolts Group folder and select Add Advanced Bolts.

You can also use “Duplicate” on an existing Advanced Bolt and modify the properties.

Inputs to define the Advanced Bolts are:

- **Bolt Head**: One or many edges connected to planar faces from solid or surface body/bodies. It can also be one Named Selection.
- **Nut/Thread**: One or many edges connected to planar faces from solid or surface body/bodies (Nut) Or One or many cylindrical faces from solid body/bodies (Thread). It can also be one Named Selection.
- **Bolt Geometry File**: Select geometry file from the list. Default “FlangeBolt_ISO15071”.
- **Material**: Select material name from the list of available materials from Engineering Data.
- **Bolt Dimension M**: Nominal bolt dimension in unit mm.
- **Head Diameter**: Read only parameter showing the head diameter.
- **Bolt Length**: Total length of bolt shaft including thread. Default 0.0 will make the bolt long enough to fill the thread selection.
- **Head friction**: Friction coefficient used in the head contact, default 0.14.
- **Thread friction**: Friction coefficient used in the thread contact, default 0.14.

\[ \mu = 0: \text{No Separation}, \mu = 1: \text{Bonded Always}, 0 < \mu < 1 \text{ Friction contact.} \]

Note that the contact thread section will have no effect if bonded contact is used.

- **Pretension Torque**: Applied bolt pretension torque used to calculate pretension force.
- **Tightening factor** \( \alpha_A \): Factor to divide the nominal bolt pretension force to account for the scatter of the achievable assembly preload from variation in friction and torque.

\[ \alpha_A = \frac{F_{M_{\text{max}}}}{F_{M_{\text{min}}}} \geq 1 \text{ and } F_{\text{pretension}} = \frac{F_{M_{\text{max}}}}{\alpha_A}. \]

The scatter amplitude is defined as: \( (F_{M_{\text{max}}} - F_{M_{\text{min}}})/(F_{M_{\text{max}}} + F_{M_{\text{min}}}) = (\alpha_A - 1)/(\alpha_A + 1) \) The definition follows VDI 2230 section 5.4.3 [3]. Guide values for the tightening factor are found in VDI 2230 table A8 [3].

The tightening factor can also be used to account for embedding in the joint that will reduce the pretension force. See the section “Embedding calculation” for more details.

- **Pretension Force**: Force value in the section load “SLOAD” command. If zero, the pretension section is locked. Used to define the pretension force direct and calculate the equivalent pretension torque based on friction coefficients and tightening factor.
- **Load Step Apply**: Load step to apply the force. Used only if Pretension > 0.
- **Load Step Lock**: Load step to lock the force to a displacement. Used only if Pretension > 0.
Bolt Geometry

The geometry is based on tabulated values (in mm) from a csv file found in the extension installation folder. Default tables for flange bolt and socket head cap bolts are included in the installation.

Table 1. FlangeBolt_ISO15071.csv

<table>
<thead>
<tr>
<th>Dim. M</th>
<th>Pth</th>
<th>rs</th>
<th>rf</th>
<th>Tf</th>
<th>af</th>
<th>Hh</th>
<th>rh</th>
<th>dm</th>
</tr>
</thead>
<tbody>
<tr>
<td>3</td>
<td>0.5</td>
<td>1.268</td>
<td>2.905</td>
<td>0.655</td>
<td>20</td>
<td>4.161</td>
<td>1.982</td>
<td>2.680</td>
</tr>
<tr>
<td>4</td>
<td>0.7</td>
<td>1.764</td>
<td>3.856</td>
<td>0.785</td>
<td>20</td>
<td>4.796</td>
<td>2.624</td>
<td>3.550</td>
</tr>
<tr>
<td>5</td>
<td>0.8</td>
<td>2.126</td>
<td>4.700</td>
<td>1.000</td>
<td>20</td>
<td>5.600</td>
<td>3.451</td>
<td>4.480</td>
</tr>
<tr>
<td>6</td>
<td>1</td>
<td>2.529</td>
<td>5.800</td>
<td>1.100</td>
<td>20</td>
<td>6.900</td>
<td>3.960</td>
<td>5.350</td>
</tr>
<tr>
<td>7</td>
<td>1</td>
<td>3.028</td>
<td>6.625</td>
<td>1.150</td>
<td>20</td>
<td>7.503</td>
<td>4.547</td>
<td>6.350</td>
</tr>
<tr>
<td>8</td>
<td>1.25</td>
<td>3.413</td>
<td>7.450</td>
<td>1.200</td>
<td>20</td>
<td>8.500</td>
<td>4.979</td>
<td>7.190</td>
</tr>
<tr>
<td>12</td>
<td>1.75</td>
<td>5.177</td>
<td>11.250</td>
<td>1.800</td>
<td>20</td>
<td>12.100</td>
<td>7.502</td>
<td>10.860</td>
</tr>
<tr>
<td>16</td>
<td>2</td>
<td>7.069</td>
<td>15.300</td>
<td>2.400</td>
<td>20</td>
<td>15.200</td>
<td>10.526</td>
<td>14.700</td>
</tr>
<tr>
<td>18</td>
<td>2.5</td>
<td>7.818</td>
<td>17.158</td>
<td>2.602</td>
<td>20</td>
<td>16.695</td>
<td>11.600</td>
<td>16.380</td>
</tr>
<tr>
<td>20</td>
<td>2.5</td>
<td>8.381</td>
<td>19.059</td>
<td>2.862</td>
<td>20</td>
<td>18.366</td>
<td>12.883</td>
<td>18.380</td>
</tr>
<tr>
<td>22</td>
<td>2.5</td>
<td>9.821</td>
<td>20.959</td>
<td>3.121</td>
<td>20</td>
<td>20.037</td>
<td>14.165</td>
<td>20.380</td>
</tr>
<tr>
<td>24</td>
<td>3</td>
<td>10.585</td>
<td>22.860</td>
<td>3.381</td>
<td>20</td>
<td>21.708</td>
<td>15.448</td>
<td>22.050</td>
</tr>
<tr>
<td>36</td>
<td>4</td>
<td>16.126</td>
<td>34.262</td>
<td>4.939</td>
<td>20</td>
<td>31.735</td>
<td>23.142</td>
<td>33.400</td>
</tr>
<tr>
<td>39</td>
<td>4</td>
<td>17.626</td>
<td>37.112</td>
<td>5.328</td>
<td>20</td>
<td>34.242</td>
<td>25.066</td>
<td>36.400</td>
</tr>
</tbody>
</table>

Pth | Thread Pitch
rs | Shaft radius (stress area)
rf | Flange outer contact radius
Tf | Flange thickness at contact radius
af | Flange angle [deg]
Hh | Total head height (including Tf)
rh | Average head radius (same area as for the hexagon part)
dm | Average thread diameter (used in pretension calculation)

Note: The thread outer diameter is based on the selected cylindrical face.

Table 2. HexSocketHead_ISO4762.csv (right part of figure)

<table>
<thead>
<tr>
<th>Dim. M</th>
<th>Pth</th>
<th>rs</th>
<th>rf</th>
<th>Tf</th>
<th>af</th>
<th>Hh</th>
<th>rh</th>
<th>dm</th>
</tr>
</thead>
<tbody>
<tr>
<td>2</td>
<td>0.5</td>
<td>1</td>
<td>1.9</td>
<td>0.503</td>
<td>0</td>
<td>2</td>
<td>-1.9</td>
<td>1.605</td>
</tr>
<tr>
<td>3</td>
<td>0.5</td>
<td>1.5</td>
<td>2.75</td>
<td>0.766</td>
<td>0</td>
<td>3</td>
<td>-2.75</td>
<td>2.680</td>
</tr>
<tr>
<td>4</td>
<td>0.7</td>
<td>2</td>
<td>3.5</td>
<td>1.000</td>
<td>0</td>
<td>4</td>
<td>-3.5</td>
<td>3.550</td>
</tr>
<tr>
<td>5</td>
<td>0.8</td>
<td>2.5</td>
<td>4.25</td>
<td>1.238</td>
<td>0</td>
<td>5</td>
<td>-4.25</td>
<td>4.480</td>
</tr>
<tr>
<td>6</td>
<td>1</td>
<td>3</td>
<td>5</td>
<td>1.479</td>
<td>0</td>
<td>6</td>
<td>-5</td>
<td>5.350</td>
</tr>
<tr>
<td>8</td>
<td>1.25</td>
<td>4</td>
<td>6.5</td>
<td>1.98</td>
<td>0</td>
<td>8</td>
<td>-6.5</td>
<td>7.190</td>
</tr>
<tr>
<td>12</td>
<td>1.75</td>
<td>5</td>
<td>8</td>
<td>2.470</td>
<td>0</td>
<td>10</td>
<td>-8</td>
<td>9.030</td>
</tr>
<tr>
<td>14</td>
<td>2</td>
<td>7</td>
<td>10.5</td>
<td>3.330</td>
<td>0</td>
<td>14</td>
<td>-10.5</td>
<td>12.700</td>
</tr>
<tr>
<td>16</td>
<td>2</td>
<td>8</td>
<td>12</td>
<td>3.862</td>
<td>0</td>
<td>16</td>
<td>-12</td>
<td>14.700</td>
</tr>
<tr>
<td>18</td>
<td>2.5</td>
<td>9</td>
<td>13.5</td>
<td>4.405</td>
<td>0</td>
<td>18</td>
<td>-13.5</td>
<td>16.380</td>
</tr>
<tr>
<td>20</td>
<td>2.5</td>
<td>10</td>
<td>15</td>
<td>4.960</td>
<td>0</td>
<td>20</td>
<td>-15</td>
<td>18.380</td>
</tr>
<tr>
<td>22</td>
<td>2.5</td>
<td>11</td>
<td>16.5</td>
<td>5.526</td>
<td>0</td>
<td>22</td>
<td>-16.5</td>
<td>20.380</td>
</tr>
<tr>
<td>24</td>
<td>3</td>
<td>12</td>
<td>18</td>
<td>6.104</td>
<td>0</td>
<td>24</td>
<td>-18</td>
<td>22.050</td>
</tr>
<tr>
<td>27</td>
<td>3</td>
<td>13.5</td>
<td>20</td>
<td>6.892</td>
<td>0</td>
<td>27</td>
<td>-20</td>
<td>25.050</td>
</tr>
<tr>
<td>30</td>
<td>3.5</td>
<td>15</td>
<td>22.5</td>
<td>7.906</td>
<td>0</td>
<td>30</td>
<td>-22.5</td>
<td>27.730</td>
</tr>
<tr>
<td>33</td>
<td>3.5</td>
<td>16.5</td>
<td>25</td>
<td>8.952</td>
<td>0</td>
<td>33</td>
<td>-25</td>
<td>30.730</td>
</tr>
<tr>
<td>36</td>
<td>4</td>
<td>18</td>
<td>27</td>
<td>9.811</td>
<td>0</td>
<td>36</td>
<td>-27</td>
<td>33.400</td>
</tr>
<tr>
<td>42</td>
<td>4.5</td>
<td>21</td>
<td>31.5</td>
<td>11.820</td>
<td>0</td>
<td>42</td>
<td>-31.5</td>
<td>38.985</td>
</tr>
<tr>
<td>48</td>
<td>5</td>
<td>24</td>
<td>36</td>
<td>13.931</td>
<td>0</td>
<td>48</td>
<td>-36</td>
<td>44.592</td>
</tr>
</tbody>
</table>

Note: Combinations of flange/washer and socket head cap is possible (left part of figure)
Custom Bolt Geometry

The user can create custom bolt tables by following these steps:

- Open the default file (FlangeBolt_ISO15071.csv) in e.g. Excel.
- Edit the values according to your specifications. Unit must be in mm and degrees.
  Note that the relation \( rf > abs(rh) > rs \) must be valid otherwise the mesh will be corrupt.
  If \( rf = rh \) the flange is removed. If \( rh < 0 \) a hex socket bolt head is made.
  The socket min depth, \( t \), and socket radius, \( r_{\text{sock}} \), are based on the head radius, \( rh \), according to:
  \[ t = 0.0051rh^2 - 0.5943rh \] \( r_{\text{sock}} = -0.5281rh \).
  (Based on ISO4762 geometry)
- Save the table as a csv-file in the extension installation folder for the current Ansys version.

 Pretension calculation

The bolt pretension is calculated based on the bolt geometry details given in the bolt geometry file (Pitch and average thread diameter), the bolt hole diameter and the bolt torque and friction details. The relation between the pretension torque, \( M \), and pretension force \( F_M \) is defined by the following equation:

\[
M = \alpha_A F_M \left( \frac{d_2}{2} \cdot \frac{P}{(\pi d_2)} + 1.155\mu_G \right) + \mu_H d_H/2 = \alpha_A F_M (0.1591P + 0.5775d_2\mu_G + 0.5\mu_H d_H)
\]

\( \alpha_A \) | Pretension factor  
\( F_M \) | Bolt pretension force  
\( d_2 \) | Average thread diameter  
\( P \) | Bolt pitch  
\( \mu_G \) | Thread friction coefficient  
\( \mu_H \) | Head friction coefficient  
\( d_H \) | Average head contact diameter

 Embedding calculation

The bolt pretension is usually reduced due to relaxation of the bolt and local plastic deformation in the bolt head and thread contacts as well as the joint contacts. If the analysis is based on linear elastic material properties, these relaxation effects can be included by reducing the nominal bolt elongation with the embedding from the different contact zones. Common values for embedding are found in VDI 2230 table 5.4/1, see below. The total sum of embedding for a bolt joint is: \( \delta_{\text{embedd}} \).

To calculate the reduced bolt pretension due to embedding a nominal pretension analysis is made first where the total bolt joint elongation, \( \delta_{\text{Adj}} \), is extracted. Use the Bolt Group Result to list the elongation or “Adjustment”.

The tightening factor is defined as: \( \alpha_A = \delta_{\text{Adj}}/(\delta_{\text{Adj}} - \delta_{\text{embedd}}) \)

<table>
<thead>
<tr>
<th>Average roughness height Rz according to DIN 4768</th>
<th>Loading</th>
<th>Guide values for amounts of embedding [( \mu m )]</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>in the thread</td>
<td>per head or nut bearing area</td>
<td>per inner interface</td>
</tr>
<tr>
<td>&lt; 10 ( \mu m )</td>
<td>Tension/compression</td>
<td>3</td>
<td>2.5</td>
</tr>
<tr>
<td></td>
<td>Shear</td>
<td>3</td>
<td>3</td>
</tr>
<tr>
<td>10 ( \mu m ) up to 40 ( \mu m )</td>
<td>Tension/compression</td>
<td>3</td>
<td>3</td>
</tr>
<tr>
<td></td>
<td>Shear</td>
<td>3</td>
<td>4.5</td>
</tr>
<tr>
<td>40 ( \mu m ) up to 160 ( \mu m )</td>
<td>Tension/compression</td>
<td>3</td>
<td>4</td>
</tr>
<tr>
<td></td>
<td>Shear</td>
<td>3</td>
<td>6.5</td>
</tr>
</tbody>
</table>
**Graphics**
The graphics in Mechanical is the same as for the **Simplified bolt**.

**Meshing recommendation**
The Advanced Bolt Group uses frictional contacts as default, and in order for this to work, especially at the thread, some detailed mesh settings are needed. The bolt thread hole should have 16 elements around in order to match the bolt shaft and make the thread contact section work. Use also a finer mesh in the axial direction since the thread forces are active at the first few thread loops. The bolt head (or nut) contact area should have inflation with two layers using first thickness of e.g. 2 mm to make a uniform stiffness around the bolt.

![Mesh setup Advanced bolt](image1)
![Von-Mises stress](image2)

It is recommended to use solid model (including solid shell) when using the Advanced Bolt but it can be used on shell models if not using the thread option. If using a shell model make sure that the surface normal point towards the bolt head contact face (or bolt nut), see figure below, otherwise the contact will fail.
**Results**

Post processing of the advanced bolts can be done using the **Bolt Group Results** object.

For bolt contact post processing MAPDL must be used by linking it to the **Solution** cell on the Workbench Project Page. You may check the “Save MAPDL db” in **Analysis Settings** in order to resume the full data base including all properties. (Commands: “PLNSOL,CONT,PRES” & “PLNSOL,CONT,STAT”)

Use Scoping Method “Solver Component (Beta)” to plot structural results (displacement and stress). Note: There is a graphical bug when displaying the solver mesh but by adding a cut plane anywhere the results are displayed correct.

**Known Issues/limitations**

The pretension can only use the **Force** and **Lock** feature of the SLOAD command and not the **Displacement**.

Drag and drop of an Advanced Bolt Group object or individual bolts between analyses is not supported. You may duplicate an analysis system and you may duplicate an individual bolt inside an Advanced Bolt Group.

The **Advanced Bolt Group** can be used in a pre-stressed **Modal** analysis by using **Advanced Bolt** in the **Static Structural** analysis. It is not enough to link the model cell from the static analysis to the **Modal** analysis. You must also link the **Solution** cell from the **Static** analysis to the **Setup** cell of the **Modal** analysis since the bolt is treated as a “load object”.

A downstream modal superposition analysis, e.g. **Harmonic** analysis, cannot use surface based loads applied by “Surface Effect”, use “Direct” instead. (“Direct FE” and “Remote point” loads work).
**Bolt Group Result**

**Background**
Post processing bolt joints is important in many applications in order to optimize the performance. This requires many manual steps in order to extract the forces and moments and calculate the utilization factors according to design codes.

**Solution**
The **Bolt Group Result** object adds the post processing features to the **Bolt Group Pretension, Simplified** and **Advanced Bolt Group** load objects. This result object will extract all relevant information from the selected bolt joint group and create a results summary listing. It also includes the Eurocode 3 [2] and VDI 2230 [3] evaluation of the bolt results.

**Usage**
Inputs to the **Bolt Group Result** are:

- **Geometry**: All Bodies (default) or selection by scooping or named selection (*).
- **Bolt Group**: A list box of all active bolt pretension group objects.
- **Bolt Class**: A list box of the desired bolt class. (Default 8.8)
- **Bolt Result**: A list box of results to display (Default “Normal Force”)
  - Normal Force/Shear Force/Bending Moment/Eurocode 3/VDI 2230
- **By**: Time/Result Set/Maximum Over Time/Time Of Maximum (standard Mechanical feature)
  - Note that you may use Maximum Over Time to get the worst case from all steps. In addition, the Graph displays the time history of min and max for the selected result.
- **Display Time**: Load step time for results evaluation. You may only use time steps where there is results saved, see the graph and tabular data window for load steps to use.

(*) The node numbers from the selected bolt group will be compared with the node numbers in the selection and results will only be plotted for nodes/bolts that are scooped in the selection.

Note that “Nodal Forces” must be set to “Yes” in **Analysis Settings** under **Output Controls** in order for post processing to work. When adding the bolt this setting is automatically set to “Yes”.

- 22 -
Output

The selected “Bolt result” at the pretension section is displayed on the geometry. For a Bolt Group Pretension object, the result is displayed on the bolt shaft for each bolt. For a “Simple” or “Advanced” bolt group the results are presented at the head contact for each bolt.

The Bolt Group Result object also writes a results listing to the solver files directory of the analysis. The naming of the output file is based on the selected Bolt Group Result name and an additional internal unique id number in case duplicate names, e.g. “Advanced_Bolt_thread_Id27.lis”. This listing includes information about all results sets, the pretension node, forces and moments and the resulting stresses. An evaluation according to Eurocode 3 [2] (utilization) and VDI 2230 [3] (safety factor) are also done.

The bolt forces are extracted at three locations along the bolt:

1. At the head section
2. At the pretension section
3. At the beginning of the thread or at the nut section.

In the output listing there are three lines for each bolt that are identified by the internal pretension node number and the location of the section in global coordinates.

Note: The only section force that can vary along the bolt is the bending moment and this will not influence the Eurocode 3 or VDI 2230 calculations unless the bolt diameter differ at the head and thread section.

The header shows the name of the analysis system (Static Structural) and the name of the bolt load object (Advanced Bolts nut). The columns of the file contains the following information:

| Set: | Result set number. See the Tabular data window for Set number and corresponding Time |
| Sect: | Section for results, 1: Head, 2: Pretension section, 3: Beginning of thread or nut |
| Pretension node id: | Master node of the bolt pretension element “PRETS179” |
| X, Y, Z: | Global coordinates of the bolt section |
| Adjust: | The locked elongation of the bolt pretension element |
| Length: | Bolt shaft free length |
| Diam: | Diameter of bolt section (stress area) |
| FX shear: | Local shear force in X direction |
| FY shear: | Local shear force in Y direction |
| FZ normal: | Normal force in the bolt section |
| MX transv: | Local translational (bending) moment around X axis |
| MY transv: | Local translational (bending) moment around Y axis |
| MZ axial: | Axial torsion moment in the bolt section excluding nominal pretension moment |
| S normal: | Maximum normal stress in bolt section, $S_{normal} = F_{normal}/A + M_{trans}/W$ |
| $A = \pi*Diam^2/4$, $W = \pi*Diam^3/32$ |
| S shear: | Maximum shear stress in bolt section, $S_{shear} = F_{shear}/A + M_{axial}/W_s$, $W_s = \pi*Diam^3/16$ |
| Bolt Class: | Selected bolt class in Bolt Group Results properties |
| EC 3: | Eurocode 3 Utilization factor (should be less than 1.0) |
| VDI 2230: | VDI 2230 Safety Factor (should be greater than 1.0) |

Notes on Output

To speed up post processing of the bolts the bolt results and properties are exported during solution of the model. In the solution folder you will find a bolt data text file, e.g. “Advanced_Bolt_Id27.bolt” that contains FE-model data for the selected bolt group and a results listing, e.g. “boltRes_Id27.lis”, in solution units.

The file “boltdata_header.csv” contains the header line with information about the columns in the bolt data file; “# Pret node,X,Y,Z,NX,NY,NZ,pretType,pretSect,boltMat” and the file “boltRes_header.csv” contains the header line for the solution listing;

“# Nset,Sect,Pret node,X,Y,Z,Adjust,Length,Diam,FX,FY,FZ,MX,MY,MZ”.
**Eurocode 3**

The Eurocode 3 [2] calculates the bolt utilization for combined shear and tension. The bolt utilization value must be lower than 1.0 in order to fulfil the requirement. Bolt bearing resistance and base material are not evaluated. Reference to sections in Eurocode 3 are given in between brackets.

Shear resistance per shear plane:  \( F_{v,Rd} = (\alpha_v f_{ub} A) / \gamma_{M2} \)  

<table>
<thead>
<tr>
<th>Bolt class</th>
<th>( \alpha_v )</th>
</tr>
</thead>
<tbody>
<tr>
<td>4.6, 5.6 and 8.8</td>
<td>0.6</td>
</tr>
<tr>
<td>4.8, 5.8, 6.8 and 10.9</td>
<td>0.5</td>
</tr>
</tbody>
</table>

Bolt ultimate tensile strength: \( f_{ub} \) E.g. bolt class 8.8 has \( f_{ub} = 800 \) MPa. (Table 3.1)

Bolt shaft section area:  \( A \)

Partial safety factor:  \( \gamma_{M2} = 1.25 \) (Can be edited in the result details) (Table 2.1)

Tension resistance:  \( F_{t,Rd} = (k_2 f_{ub} A_s) / \gamma_{M2} \)  

<table>
<thead>
<tr>
<th>Bolt class</th>
<th>( k_2 )</th>
</tr>
</thead>
<tbody>
<tr>
<td>4.6, 5.6 and 8.8</td>
<td>0.9</td>
</tr>
<tr>
<td>4.8, 5.8, 6.8 and 10.9</td>
<td>0.9</td>
</tr>
</tbody>
</table>

Combined shear and tension:  \( F_{v,Ed} / F_{v,Rd} + F_{t,Ed} / (1.4 F_{t,Rd}) \leq 1.0 \)  

Pre-stressed bolts must always be of class 8.8 or higher. If the user selects a lower bolt class together with Eurocode 3 a warning message will be issued and the utilization will be calculated with bolt quality 8.8. If result selection is Eurocode 3 an info message will be issued if the applied pretension force is lower than the recommended pretension force.

Recommended bolt pretension force:  \( F_{p,Cd} = (0.7 f_{ub} A_s) / \gamma_{M7} \)  

Preload of high strength bolts:  \( \gamma_{M7} = 1.1 \)

**VDI 2230**

The VDI 2230 [3] calculates the bolt safety factor based on an equivalent stress from the normal force and torsion moment from the bolt pretension. Reference to VDI 2230 Part 1 are given in between brackets.

Maximum tensile stress:  \( \sigma_{t,\text{max}} = F_{S,\text{max}} / A_0 \)  

Bolt shaft normal force:  \( F_{S,\text{max}} \)

Bolt shaft section stress area:  \( A_0 \)

Maximum Torsional stress:  \( \tau_{\text{max}} = M_G / W_p \)  

\[
M_G = F_{M,\text{zul}} \times (d_2/2) \times (P/(\pi d_2) + 1.155 \mu_{G,\text{min}}) = F_{M,\text{zul}} \times (0.1591 P + 0.5775 d_2 \mu_{G,\text{min}})
\]

Bolt pretension force:  \( F_{M,\text{zul}} \)

Average thread diameter:  \( d_2 \)

Bolt pitch:  \( P \)

Minimum thread friction coefficient:  \( \mu_{G,\text{min}} \)

W_p = \( \pi d^3/16 \)

Section diameter:  \( d \)

Reduced comparative stress:  \( \sigma_{\text{red},B} = \sqrt{\sigma_{t,\text{max}}^2 + 3(k_1 \tau_{\text{max}})^2} \)  

In service recommendation:  \( k_1 = 0.5 \)

Safety factor:  \( S_F = R_{p0.2,\text{min}} / \sigma_{\text{red},B} \)  

Bolt yield limit:  \( R_{p0.2,\text{min}} \) E.g. bolt class 8.8 has  \( R_{p0.2,\text{min}} = 0.8 \times 800 = 640 \) MPa.
Notes on VDI 2230 post processing

The VDI 2230 post processing calculates an equivalent stress based on the normal stress and the torsion stress from the pretension. For the “Advanced bolt” all properties are defined in order to calculate the torsion stress in the bolt shaft. For a “Multi-pretension” or “Simplified bolt” the thread friction is set to 0.10. The thread pitch, $P_{th}$, is estimated from the bolt shaft stress diameter, $D_s$, according to: $P_{th}(D_s) = 0.4795 + 0.1079 * D_s$. This formula is based on the pitch for the standard ISO 15071 flange bolt.

Also note that the color bar needs to be reversed to reflect that VDI result is a safety factor where a lower value is more critical, hence red should be at the bottom of the bar.

Notes on derived properties

From the results listing additional bolt joint post processing can be done, e.g. to study embedding influence on bolt pretension (see the section in Advanced Bolt Group) or the relation in stiffness between bolt shaft and support.

Nominal Bolt pretension force:

- $F_{nom}$
- $K_s = E A_s / L_s$
- $E$
- $A_s$
- $L_s$
- $\delta_{Adj}$
- $\delta_{bolt} = F_{nom} / K_s$
- $\delta_{embedd}$
- $\alpha_A = \delta_{Adj} / (\delta_{Adj} - \delta_{embedd})$
- $F_{red} = F_{nom} / \alpha_A$

Known Issues/limitations

The Bolt Group Result cannot use a “Bolt Group Pretension” load that is applied on wire bodies. Use the “Simplified Bolts” instead to create the connections in order to post process.
Mapping of thermal loads
If using an upstream thermal analysis with “Imported Body Temperatures” and/or “Thermal Condition” in the structural analysis these temperatures will be automatically mapped to the rivet and bolts. Temperatures will be interpolated over the bolt using the head and thread/nut area temperatures.

If using a mix of imported and thermal conditions, like in the figure below, each part of the bolt/rivet will have constant temperature based on its connected part. For a solid bolt the entire bolt will have constant temperature based on the connected parts thermal condition at the bolt head.

Detail of mapped temperatures with interpolated temperatures on the advanced bolts.
Resolve solution convergence problems for simplified bolts and rivets

Using rivets or simplified bolts will add beam elements to the FE-model. This influences the moment convergence. Using the default setting “Program Controlled” for Moment Convergence in Analysis Settings/Nonlinear Controls may result in long solution time or un-converged solutions for certain types of models and loads.

Review the Solution Output: “Force Convergence” and “Moment Convergence” for the first pretension step to find the convergence criteria value “Force Criteria” and “Moment Criteria”. If the “Moment Criteria” drops to a very low value for load step two and up and the solution struggle to converge you may assign a manual value.

Assign Moment Convergence “On” and set the “Value” and “Tolerance” for all load steps to have manual criteria. The “criteria” is defined from “Value*Tolerance”. In this case Value = 20000 and Tolerance = 1% gave a similar convergence behavior for both Force and Moment.

Tip: Click on “Analysis Settings” and then “Worksheet” button to review settings for all load steps.

<table>
<thead>
<tr>
<th>Analysis Settings</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
</tr>
<tr>
<td>Step Controls</td>
</tr>
<tr>
<td>Step End Time</td>
</tr>
<tr>
<td>Auto Time Stopping</td>
</tr>
<tr>
<td>Define By</td>
</tr>
<tr>
<td>Carry Over Time</td>
</tr>
<tr>
<td>Initial Substeps</td>
</tr>
<tr>
<td>Maximum Substeps</td>
</tr>
<tr>
<td>Nonlinear Controls</td>
</tr>
<tr>
<td>Force Convergence</td>
</tr>
<tr>
<td>Moment Convergence</td>
</tr>
<tr>
<td>Value</td>
</tr>
<tr>
<td>Tolerance</td>
</tr>
<tr>
<td>Displacement Convergence</td>
</tr>
<tr>
<td>Rotation Convergence</td>
</tr>
<tr>
<td>Linear Search</td>
</tr>
<tr>
<td>Stabilization</td>
</tr>
</tbody>
</table>
References

[1] ANSYS Help R18 online documentation
