This tutorial is intended to help first-time users become familiar with basic usage of Design+ for RC beam and column design.
Design+ Interface

**SIMPLE MODE**

User interface is very simple and intuitive so that first-time users become familiar with its interface without any difficulty.

**WORKBAR**

- Member name change, Grouping, Report and Drawing Export Setup

**INPUT**

- Member force (similar with midas Set)

**PROJECT MODE**

Drawing, Project management, Batch report generation

**Preview**

Design or check results are displayed instantly.

**REPORT**

Generate input data, summary, and detail design result report
- Detail
- Summary
- Input List

**midas Link**

Link with midas Gen
Multiple members can be selected and imported.

**Message Window**

Display various information, warning, and error messages.
Design Process of RC Beam

This tutorial is intended to help first-time user of midas Design+ by explaining general usage and design process. The user can check midas Gen link, Workbar usage, and beam design process, and design result verification.
1. Main Menu > Link > midas
2. Select midas Gen model file to be linked.
3. Click [Connect] button.
4. Confirm message window to check link status.
5. Select desired members to be imported into midas Design+.
6. Confirm member number.
7. Click [Import] button.
8. Check imported beam members in the Workbar.

Note
- When design or checking has been performed in midas Gen, the member forces can be imported in Design+.
- From the main menu, select Link > Link Option to specify various options for import.
  - [Link by Section]: The most critical design results among the members assigned with the identical section properties are imported in Design+.
  - [Link by Member]: Individual beam members are imported in Design+.
1. Click [Design] button.
2. From the main menu, go to Option > Design Option.
3. Specify the desired ratio (Design Member Force / Resistance) by design components.
4. Click [Apply] and [Close] button.
5. From the main menu, go to Option > Rebar Option.
6. Specify rebar option by member type.
7. Define the desired rebar spacing for main rebar and shear rebar.
8. Click [Apply] and [Close] button.
**Step 03 Beam Design 1 (Simple Mode)**

1. Specify material data.
2. Specify section data,
   - Enter width, height, cover concrete thickness.
   - Check on “Same Top&Bot. Cover” option to apply identical cover concrete thickness between top and bottom.
3. Specify design type.
   - Single Rebar: Singly reinforced beam design
   - Double Rebar: Doubly reinforced beam design
4. Specify section shape.
   - Rectangular
   - T Shape
5. Enter design member force.

**Note**
- When Shape is entered as “T Shape”, the user can enter the slab thickness and effective width.
- When negative value of moment is entered, the section will be designed about negative moment.

**Step 03 Beam Design 2**

1. Click Rebar tab.
2. Enter or modify rebar.
3. Enter Skin Bar.
   - Evenly distribute at side: Arrange the skin rebar to be distributed vertically.
4. Specify Splice type of main rebar.
   - No Splice
   - 50% Splice
   - 100% Splice
5. Click [Design] or [Check] button.
1. Click Project Mode.
2. Select “Type-3(Each end & Center)” option for Rebar Arrangement.
3. Check and modify rebar data.
4. Click [Design] or [Check] button.
   - When performing “Design”, the entered rebar data will be automatically updated to find the satisfied design results.
   - When performing “Check”, the program verify the member using the rebar data entered by the user.

**Rebar Arrangement**
- Type-1 (All Section): Select when rebar data for i-end, middle and j-end are identical.
- Type-2 (Both End & Center): Select when rebar data of i-end and j-end are identical.
- Type-3 (Each End & Center): Select when rebar data of i-end, middle and j-end are identical.

<table>
<thead>
<tr>
<th>Type-1</th>
<th>Type-2</th>
<th>Type-3</th>
</tr>
</thead>
<tbody>
<tr>
<td>max(l,m,j)</td>
<td>l &lt; m,j</td>
<td>l &lt; (m,j)</td>
</tr>
</tbody>
</table>

- Design a member using the maximum member forces among i-end, j-end, and Middle.
- Change rebar input data and member force between i-end and j-end.
- Change Force Only: Check on to apply this option for design member force only.

- Moment, top (kN.m): Negative moment
- Moment, bot (kN.m): Positive moment
- Shear (kN)
- Rebar, top: Number of rebar – Diameter
- Rebar, bot: Number of rebar – Diameter
- Stirrup (mm): Number of rebar – Diameter@Spacing
- Main Bar Space (mm): Maximum rebar spacing
- Skin Bar Space (mm): Skin rebar spacing
- Comment
**Beam Design 4**

1. Click Member List tab.
2. Verify member list.
3. Click [Apply] button. Once the user click [Apply] button, changes in Member List Tab will be applied in Member Tab.

**Note**
- In order to save changes in the member list table, the user needs to click [Apply] button.

**Beam Design 5**

1. Click Drawing tab.
2. Select RC and Beam in the combo box. Click [Create] button.

**Note**
- Generated drawings can be modified using general AutoCAD commands.
Step 03 Beam Design 6

1. Click Quantity tab.
2. Select RC and Beam in the combo box. Click [Create] button.
3. Check rebar data for i-end, middle, and j-end.
4. Confirm quantity per unit length for concrete, form, rebars.

Note
- Quantity in the table is shown as the rebar weight and concrete volume by unit area (1m²).

Step 04 Change member name in Workbar

1. In order to modify member name displayed in the Workbar, right-click on the desired member and select “Rename” from the context menu. “F2” key from keyboard can also be used.
   - Press [Enter] after entering the new member name.
1. In order to change the member order in Workbar, right-click on Beam and select Reordering.
2. Modify the order of members.

Note: The user can simply use "drag & drop" on the Workbar to change the member order.
# Contents

1. Preference  
2. Workbar Reordering  
3. Change Member Name in Workbar

## midas Design+ Design Parameter

<table>
<thead>
<tr>
<th>RC</th>
<th>STEEL</th>
<th>SRC</th>
</tr>
</thead>
<tbody>
<tr>
<td>Slab</td>
<td>Beam &amp; Column</td>
<td>Composite Beam</td>
</tr>
<tr>
<td>Shear Wall</td>
<td>Base Plate</td>
<td></td>
</tr>
<tr>
<td>Footing</td>
<td>Bolt Connection</td>
<td></td>
</tr>
<tr>
<td>Basement Wall</td>
<td>Crane Girder</td>
<td></td>
</tr>
<tr>
<td>Buttress</td>
<td>Purin &amp; Girth</td>
<td></td>
</tr>
<tr>
<td>Stair</td>
<td>Web Opening</td>
<td></td>
</tr>
<tr>
<td>Column</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Column (Gen)</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Beam Totio</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Slab Totio</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Batch Wall</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Copyright © Since 1989, MIDAS Information Technology Co., Ltd.
1. From the main menu, click Option > Preference. Click “RC(1)” tab.
2. Click “RC(2)” tab.
3. Click “Section” tab.

**Note**
- Preference dialog box can be invoked from Workbar by clicking “Design Option > Preference”.
- Click [Default] button to reset all the preference setting into the initial values.

**Slab / Shear Wall**
- **Change section by Design**: Check on to update section size in Design.
- **Max. Thick**: Check on to specify the maximum thickness.

**Beam / Column**
- **Change section by Design**: Check on to update section size in Design.
- **Max. Width / Max. Height**: Check on to specify the maximum width and height.

**Basement Wall / Footing**
- **Change section by Design**: Check on to modify the maximum thickness in Design.
- **Max. Thick**: Check on to specify the maximum thickness.
- **Apply shear reinforcement**: Specify the applicable shear rebar diameter.

**Buttress / Corbel**
- **Change section by Design**: Check on to update section size in Design.
- **Max. Width / Max. Height**: Check on to specify the maximum width and height.
- **Max. Layer No.**: Maximum number of layers to be used in Design.

**Stair**
- **Change section by Design**: Check on to update section size in Design.
- **Max. Thick**: Check on to specify the maximum thickness.

**Section Increment(RC)**: Specify the dimension increment for Design.
1. Click “General” tab.
2. Click “Word” tab.

User Interface Mode
• Select default model when executing the program.

Default Report Type for Simple Mode/Check Mode
• Default report type once design or check is performed.

Cover Concrete Depth
• Use clear cover: Define cover concrete depth as clear depth
• Use distance from face to rebar center: Define cover concrete depth as the distance from the center of the rebar to the concrete face
※ In case of column, “Use distance from face to rebar center” option is always applied.

Result data
• Do not Delete Results Data When Input Data is Changed: Remain design results when design parameters are changed.
• Include design result in Input File(Excel): Check on to include design results in input list excel report.
• Show “Apply” Button in Member Dialog: Display [Apply] button in Member tab.

Word
• Modify default setting of font and text size of MS word report generated from Project Mode.
1. Click Drawing tab.
2. Click File tab.
3. Click Layer tab.

**Preference Setting 3**

#### Tie bar of Circular Column
- **Parallel**:
- **Radial**:

#### Rebar Name
- **Use Name by Strength**: Check on to enter the rebar name by rebar strength.

#### Print design force

#### Frame File / Legend File
- Specify the file path of default drawing frame and legend file.

#### Layer
- Specify the default line color and line type for drawing.
### Step 02 Workbar Reordering

1. In order to change the member order in Workbar, right-click on Beam and select Reordering.
2. Modify the order of members.
   2a. Select members to change the order and click [>>] button.
   2b. Click [Up] or [Down] button to change the member order.
   2c. Click [Apply] and [OK] button.
   2d. Check the updated member order in Workbar.

**Note**
- The user can simply use “drag & drop” on the Workbar to change the member order.

### Step 03 Change member name in Workbar

1. In order to modify member name displayed in the Workbar, right-click on the desired member and select “Rename” from the context menu.
   “F2” key from keyboard can also be used.
   - Press [Enter] after entering the new member name.