STAAD HELP MANUAL

Professor
Nabeel AL-Bayati

تحليل وتصميم
المنشآت الهندسية والخرسانية باستخدام الحاسب
(الصف الرابع / المحاضرات النظرية)

2010
## STAAD-III Manual Index

<table>
<thead>
<tr>
<th>No.</th>
<th>Title</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.</td>
<td>Command Language Conventions</td>
<td>4 - 7</td>
</tr>
<tr>
<td></td>
<td>Elements of the Commands</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Command Formats</td>
<td></td>
</tr>
<tr>
<td>2.</td>
<td>Problem Initiation And Title</td>
<td>7 - 8</td>
</tr>
<tr>
<td>3.</td>
<td>Unit Specification</td>
<td>8 - 9</td>
</tr>
<tr>
<td>4.</td>
<td>Joint Coordinate Specification</td>
<td>9 - 11</td>
</tr>
<tr>
<td>5.</td>
<td>Member Incidence Specification</td>
<td>11 - 13</td>
</tr>
<tr>
<td>6.</td>
<td>Delete Specification</td>
<td>13 - 14</td>
</tr>
<tr>
<td>7.</td>
<td>Member Property Specification</td>
<td>14 - 17</td>
</tr>
<tr>
<td></td>
<td>Specifying Properties from Steel Table</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Prismatic Property Specification</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Examples of Member Property Specification</td>
<td></td>
</tr>
<tr>
<td>8.</td>
<td>Member Release Specification</td>
<td>17 - 18</td>
</tr>
<tr>
<td>9.</td>
<td>Member Truss Specification</td>
<td>18</td>
</tr>
<tr>
<td>10.</td>
<td>Constant Specification</td>
<td>19</td>
</tr>
<tr>
<td>11.</td>
<td>Global Support Specification</td>
<td>19 - 20</td>
</tr>
<tr>
<td>12.</td>
<td>Draw Specifications</td>
<td>20 - 22</td>
</tr>
<tr>
<td>13.</td>
<td>Definition of Load Systems</td>
<td>22 - 26</td>
</tr>
<tr>
<td></td>
<td>Selfweight Load Specification</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Joint Load Specification</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Member Load Specification</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Temperature Load Specification for Members</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Support Displacement Load Specification</td>
<td></td>
</tr>
<tr>
<td>14.</td>
<td>Load Combination Specification</td>
<td>26</td>
</tr>
<tr>
<td>16.</td>
<td>Load List Specification</td>
<td>28</td>
</tr>
<tr>
<td>17.</td>
<td>Section Specification</td>
<td>29 - 30</td>
</tr>
<tr>
<td>18.</td>
<td>Print Specification</td>
<td>30 - 32</td>
</tr>
<tr>
<td>19.</td>
<td>Steel Design Specifications</td>
<td>32 - 35</td>
</tr>
<tr>
<td></td>
<td>Parameter Specification</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Code Checking Specification</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Member Selection Specification</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Member Selection by Optimization</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Steel Take Off Specification</td>
<td></td>
</tr>
<tr>
<td>20.</td>
<td>Concrete Design Specifications</td>
<td>36 - 39</td>
</tr>
<tr>
<td></td>
<td>Design Initiation</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Concrete Design Parameter Specification</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Concrete Design Command</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Concrete Take Off Command</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Concrete Design Terminator</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Design Initiation</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Footing Design Parameter Specification</td>
<td></td>
</tr>
</tbody>
</table>
1-Command Language Conventions

This section describes the command language used in STAAD-III. First, the various elements of the language are discussed and then the command format is described in detail.

Elements of the Commands

a) Integer Numbers: Integer numbers are whole numbers written without a decimal point. These numbers are designated as i1, i2, etc., and should not contain any decimal point. Signs (+ or -) are permitted in front of these numbers. If the sign is omitted, it is assumed to be positive (+).

b) Floating Point Numbers: These are real numbers which may contain a decimal portion. These numbers are designated as f1, f2... etc.. Values may have a decimal point and/or exponent.

Example

5055.32 0.73 -8.9 732
5E3 -3.4E-6
etc.
When the sign is omitted, it is assumed to be positive (+). Also note that the decimal point may be omitted, if the decimal portion of the number is zero.

c) Alphanumerics: These are characters which are used to construct the names for data, titles or commands. All alphabetic characters must be input in upper case, or capital letters. No quotation marks are needed to enclose them.

Example

MEMBER PROPERTIES
1 TO 8 TABLE ST W8X35

d) Repetitive Data: Repetitive numerical data may be provided by using the following format:

n*f

where n = number of times data has to be repeated
f = numeric data, integer and floating point

Example

JOINT COORDINATES
1 3 * 0.
This joint coordinate specification is same as:

1 0 0 0.

**Command Formats**

a) Free-Format Input: All input to STAAD-III is in free-format style. Input data items should be separated by blank spaces or commas from the other input data items. Quotation marks are never needed to separate any alphabetic words such as data, commands or titles.

b) Commenting Input: For documentation of a STAAD-III data file, the facility to provide comments is available. Comments can be included by providing an asterisk (*) mark as the first non-blank character in any line. The line with the comment is "echoed" in the output file but not processed by the program.

**Example**

JOINT LOAD
* THE FOLLOWING IS AN EQUIPMENT LOAD
2 3 7 FY 35.0
etc.

c) Meaning of Underlining in the Manual: Exact command formats are described in the latter part of this section. Many words in the commands and data may be abbreviated. The full word intended is given in the command description with the portion actually required (the abbreviation) underlined. For example, if the word MEMBER is used in a command, only the portion MEMB need be input. It is clearer for others reading the output if the entire word is used, but an experienced user may desire to use the abbreviations.

d) Meaning of Braces and Parenthesis: In some command formats, braces enclose a number of choices, which are arranged vertically. One and only one of the choices can be selected. However, several of the listed choices may be selected if an asterisk (*) mark is located outside the braces.

**Example**

\[
\begin{aligned}
\text{XY} \\
\text{YZ} \\
\text{XZ}
\end{aligned}
\]

In the above example, the user must make a choice of XY or YZ or XZ.

**Example**

\[
\begin{aligned}
* \text{FX} \\
\text{FY}
\end{aligned}
\]
FZ

Here the user can choose one or all of the listing (FX, FY and FZ) in any order. Parentheses, ( ), enclosing a portion of a command indicate that the enclosed portion is optional. The presence or absence of this portion affects the meaning of the command, as is explained in the description of the particular command.

**Example**

PRINT (MEMBER) FORCES
PERFORM ANALYSIS (PRINT LOAD DATA)

In the first line, the word MEMBER may be omitted with no change of the meaning of the command. In the second line,

PRINT LOAD DATA
command may also be omitted, in which case the load data will not be printed.

e) Multiple Data Separator: Multiple data can be provided on a single line, if they are separated by a semicolon (;) character. One restriction is that consecutive commands cannot be separated by a semicolon. They must appear on separate lines.

**Example**

MEMBER INCIDENCES
1 1 2; 2 2 3; 3 3 4
etc.

Possible Error:
PRINT FORCES; PRINT STRESSES

In the above case, only the PRINT FORCES command is processed and the PRINT STRESSES command is ignored.

f) Listing Data: In some STAAD-III command descriptions, the word "list" is used to identify a list of joints, members/elements or loading cases. The format of a list can be defined as follows:

\[
\text{list} = \left\{ \begin{array}{c}
i_1, i_2, i_3, \ldots \\
i_1 \text{ TO } i_2 \text{ (BY } i_3) \\
X \text{ or } Y \text{ or } Z
\end{array} \right\}
\]

TO means all integers from the first (i1) to the second (i2) inclusive. BY means that the numbers are incremented by an amount equal to the third data item (i3). If BY i3 is omitted, the increment will be set to one. Sometimes the list may be too long to fit on one line, in which case the list may be continued to the next line by providing a hyphen preceded by a blank. Also note that only a list may be continued and not any other type of data.
Instead of a numerical list, the specification X (or Y or Z) may be used. This specification will include all MEMBERs parallel to the global direction specified. Note that this is not applicable to JOINTs or ELEMENTs.

**Example**

```
2 4 7 TO 13 BY 2 19 TO 22 -
28 31 TO 33 FX 10.0
```

This list of items is the same as:

```
2 4 7 9 11 13 19 20 21 22 28 31 32 33 FX 10.0
```

Possible Error:

```
3 5 TO 9 11 15 -
FX 10.0
```

In this case, the continuation mark for list items is used when list items are not continued. This will result in an error message or possibly unpredictable results.

### 2- Problem Initiation And Title

**Purpose**

This command initiates the STAAD run, allows the user to specify the type of the structure and an optional title.

**General format:**

```
STAAD { PLANE SPACE TRUSS FLOOR } (any title a1)
```

**Description**

Any STAAD-III input has to start with the word STAAD. Following type specifications are available:

- **PLANE** = Plane frame structure
- **SPACE** = Space frame structure
- **TRUSS** = Plane or space truss structure
- **FLOOR** = Floor structure

**Notes**

*a1 = Any title for the problem. This title will appear on the top of every output page. To include additional information in the page header, use a comment line containing the pertinent information as the second line of input.*
1) The user should be careful about choosing the type of the structure. The choice is dependent on the various degrees of freedom that need to be considered in the analysis. The following figure illustrates the degrees of freedoms considered in the various type specifications.

![Structure Types](image)

2) The optional title provided by the user is printed on top of every page of the output. The user can use this facility to customize his output.

### 3- Unit Specification

**Purpose**

This command allows the user to specify or change length and force units for input and output.

**General format:**

```
UNIT
  * length-unit
  force-unit
```

- **length-unit** = \{ INCHES, FEET or FT, CM, METER, MMS, DME, KM \}
- **force-unit** = \{ KIP, POUND, KG, MTON, NEWTON, KNS, MNS, DNS \}

**Note:**

DME denotes Decameters. MNS denotes mega Newtons and DNS denotes decaNewtons. All other units are self explanatory.

**Description**

The UNIT command can be specified any number of times during an analysis. All data is assumed to be in the most recent unit specification preceding that data. Also note that the
input-unit for angles is always degrees. However, the output unit for joint rotations (in joint displacement) is radians. For all output, the units are clearly specified by the program.

Example
UNIT KIP FT
UNIT INCH
UNIT CM MTON

Notes
This command may be used as frequently as needed to specify data or generate output in the desired length and/or force units. Note that mix and match between different unit systems (Imperial, Metric, SI etc.) are allowed.

4- Joint Coordinates Specification

Purpose
This set of commands allow the user to specify and generate the coordinates of the JOINTs of the structure. The JOINT COORDINATES command initiates the specification of the coordinates. The REPEAT and REPEAT ALL commands allow easy generation of coordinates using repetitive patterns.

General format:

**JOINT** Coordinates
i1, x1, y1, z1, (i2, x2, y2, z2, i3)

**REPEAT** n, xi1, yi1, zi1, (xi2, yi2, zi2,..., xin, yin, zin)

**REPEAT ALL** n, xi1, yi1, zi1, (xi2, yi2, zi2,..., xin, yin, zin)

Description
The command JOINT COORDINATES specifies a Cartesian Coordinate System (see figure 2.2). Joints are defined using the global X, Y and Z coordinates. The command JOINT

Example

JOINT COORDINATES NOREDUCE BAND

The REPEAT command causes the previous line of input to be repeated n number of times with specified coordinate increments. The REPEAT ALL command functions similar to the REPEAT command except that it repeats all previously specified input back to the most recent REPEAT ALL command, or all joint data if no previous REPEAT ALL command has been given. (When using the REPEAT and REPEAT ALL commands, joint numbering must be consecutive and should begin with 1.)

i1 = The joint number for which the coordinates are provided. Any integer number (five digit max.) is permitted.

x1, y1 and z1 = X, Y & Z coordinates of the joint.
For PLANE analyses z1 is an optional data item when defining input for individual joints. z1 is always required for joint generation. The following are used only if joints are to be generated.

\(i2 = \) The second joint number to which the joint coordinates are generated.
\(x2, y2, \) and \(z2 = \) \(X, Y \) & \(Z \) (R, & \(Z \) for cylindrical or R, \(Y \) & \( \) for cylindrical reverse) coordinates of the joint.
\(i3 = \) Joint number increment by which the generated joints will be incremented. Defaults to 1 if left out.
\(n = \) Number of times repeat is to be carried out.
\(xik, yik \& zik = \) \(X, Y \& Z \) coordinate increments for k\th repeat.

The \(X, Y\) and \(Z\) coordinates will be equally spaced between \(i1\) and \(i2\).

**Example 1**

**JOINT COORDINATES**

1 10.5 2.0 8.5
2 0.0 0.0 0.0
3 5.25 0.0 8.5
6 50.25 0.0 8.5

In this example, \(X\ Y\ Z\) coordinates of joints 1 to 6 are provided. Note that the joints between 3 & 6 will be generated with joints equally spaced from 3 to 6. Hence, joint 4 will have coordinates of 20.25 0.0 8.5 and joint 5 will have coordinates of 35.25 0.0 8.5.

**Example 2**

**JOINT COORDINATES**

1 0.0 0.0 0.0
4 45.0 0.0 0.0

**REPEAT**

4 0.0 0.0 15.0

**REPEAT ALL**

10 0.0 10.0 0.0

Here, the 220 joint coordinates of a ten story 3 X 4-bay structure are generated. The **REPEAT command** repeats the first input line 4 times, incrementing each \(Z\) coordinate by 15. Thus, the first 2 lines are sufficient to create a "floor" of twenty joints:

\[
\begin{align*}
1 & : 0.0.0.0.; 2 & : 15.0.0.0.; 3 & : 30.0.0.0.; 4 & : 45.0.0.0. \\
5 & : 0.0.15.; 6 & : 15.0.15.; 7 & : 30.0.15.; 8 & : 45.0.15. \\
17 & : 0.60.; 18 & : 15.0.60.; 19 & : 30.0.60.; 20 & : 45.0.60.
\end{align*}
\]

The **REPEAT ALL command** repeats all previous data (i.e. the 20 joint "floor") ten times, incrementing the \(Y\) coordinate by 10 each time. This creates the 200 remaining joints of the structure:

**Example 3**

21 0.0 10.0 0.0 ; 22 15.0 10.0 0.0 ; ...
40 45.0 10.0 60.0 ; 41 0.0 20.0 0.0 ; ...
200 45.0 90.0 60.0 ; 201 0.0 100.0 0.0 ; ...
The following examples illustrate various uses of the REPEAT command.

REPEAT 10  5. 10. 5.

The above REPEAT command will repeat the last input line 10 times using the same set of increments (i.e. x = 5., y = 10., z = 5.)

REPEAT 3  2. 10. 5. 3. 15. 3. 5. 20. 3.

The above REPEAT command will repeat the last input line three times. Each repeat operation will use a different increment set.

REPEAT 10  0. 12. 0. 15*0 0. 10. 0. 9*0

The above REPEAT command will repeat the last input line 10 times; six times using x, y and z increments of 0., 12. and 0., and four times using increments of 0., 10. and 0. Each x, y and z value of 0 represents no change from the previous increment. To create the 2nd through 6th repeats, five sets of 0., 0. and 0. (15*0) are supplied. The seventh repeat is done with increments of 0., 10. and 0. The 8th through 10th repeats are done with the same increments as 7, and is represented as 9*0.

Notes
The PRINT JOINT COORDINATE command may be used to verify the joint coordinates provided or generated by REPEAT and REPEAT ALL commands.

Also, use the STAAD-POST facility to verify geometry graphically.

5- Member Incidences Specification

Purpose
This set of commands are used to specify MEMBERs by defining connectivity between JOINTs. REPEAT and REPEAT ALL commands are available to facilitate generation of repetitive patterns.

The member/element incidences must be defined such that the model developed represents one single structure only, not two or more separate structures. STAAD-III is capable of detecting multiple structures automatically.

General format:

MEM  BER  INC  IDENCES
i1, i2, i3, ( i4, i5, i6 )

REPEAT   n, mi, ji

REPEAT ALL   n, mi, ji

Description
The REPEAT command causes the previous line of input to be repeated n number of times with specified member and joint increments. The REPEAT ALL command
functions similar to the REPEAT command except that it repeats all previously specified input back to the most recent REPEAT ALL command or to the beginning of the specification if no previous REPEAT ALL command has been issued. (When using REPEAT and REPEAT ALL commands, member numbering must be consecutive).

i1 = Member number for which incidences are provided. Any integer number (maximum six digits) is permitted.
i2 = Start joint number.
i3 = End joint number.

The following data are used for member generation only:

i4 = Second member number to which members will be generated.
i5 = Member number increment for generation.
i6 = Joint number increment which will be added to the incident joints. (i5 and i6 will default to 1 if left out.)
n = Number of times repeat is to be carried out.
m = Member number increment
j = Joint number increment

Example

MEMBER INCIDENCES
1 1 2
2 5 7 5
7 11 13 13 2 3

In this example, member 1 goes from joint 1 to 2. Member 2 is connected between joints 5 and 7. Member numbers from 3 to 5 will be generated with a member number increment of 1 and a joint number increment 1 (by default). That is, member 3 goes from 6 to 8, member 4 from 7 to 9, member 5 from 8 to 10. Similarly, in the next line, member 9 will be from 14 to 16, 11 from 17 to 19 and 13 from 20 to 22.

Additional example

MEMBER INCIDENCES
1 1 21 20
21 21 22 23
REPEAT 4 3 4
36 21 25 39
REPEAT 3 4 4
REPEAT ALL 9 51 20

This example creates the 510 members of a ten story 3 X 4-bay structure (this is a continuation of the example started in Section 6.12). The first input line creates the twenty columns of the first floor:
The two commands (21 21 22 23 and REPEAT 4 3 4) create 15 members which are the second floor "floor" beams running, for example, in the east-west direction:

21 21 22; 22 22 23; 23 23 24
24 25 26; 25 26 27; 26 27 28
...  ...  ... 
33 37 38; 34 38 39; 35 39 40

The next two commands (36 21 25 39 and REPEAT 3 4 4) function similar to the previous two commands, but here create the 16 second floor "floor" beams running in the north-south direction:

36 21 25; 37 22 26; 38 23 27; 39 24 28
40 25 29; 41 26 30; 42 27 31; 43 28 32
...  ...  ...  ...
48 33 37; 49 34 38; 50 35 39; 51 36 40

The preceding commands have created a single floor unit of both beams and columns a total of 51 members. The REPEAT ALL now repeats this unit nine times, generating 459 new members and finishing the ten story structure. The member number is incremented by 51 (the number of members in a repeating unit) and the joint number is incremented by 20, (the number of joints on one floor).

Notes
The PRINT MEMBER INFO command may be used to verify the member incidences provided or generated by REPEAT and REPEAT ALL commands.

Also, use the STAAD-POST facility to verify geometry graphically.

6- Delete Specification

Purpose
This commands may be used to permanently DELETE specified JOINTs or MEMBERs.

General format:

DELET

MEMBERS  member-list

JOINTS  joint-list

Description
The DELETE command will completely delete the members from the structure; the user cannot re-activate them. These commands must be provided immediately after all member/element incidences are provided.
Notes
a) The DELETE MEMBER command will automatically delete all joints associated with deleted members, provided the joints are not connected by any other active members or elements.
b) This command will also delete all the joints which were not connected to the structure in the first place. For example, such joints may have been generated for ease of input of joint coordinates and were intended to be deleted. Hence, if a DELETE MEMBER command is used, a DELETE JOINT command should not be used.
c) The DELETE MEMBER command is applicable for deletion of members as well as elements. If the list of members to be deleted extends beyond one line, it should be continued on to the next line by providing a blank space followed by a hyphen (-) at the end of the current line. In other words, the DELETE MEMBER command can be defined only once.

Example

DELETE MEMBERS 29 TO 34 43

7- Member Property Specification

Purpose
This set of commands may be used for specification of section properties for frame members.

General format:

```
MEMBER PROPERTIES

AUSTRALIAN
CANADIAN
EUROPEAN
FRENCH
INDIAN
AMERICAN
BRITISH
GERMAN
JAPANESE
```

member - list

```
TABLE type-spec. table-name (additional-spec)
PRISMATIC property-spec
```

AMERICAN, BRITISH, EUROPEAN (etc.) option will cause the program to pick up properties from the appropriate steel table. The default depends on the country of distribution.

Description
This command initiates the specification of MEMBER PROPERTY. Following are the various options available:

a) Specification from built-in steel tables.
b) Specification of prismatic properties.

**Specifying properties from steel table**

**Purpose**
The following commands are used for specifying section properties from built-in steel table(s).

**General format:**

type-spec . table-name ( additional-spec. )

```
{ ST  RA  
  D   LD  
  SD  T   
  CM  TC  
  BC  TB  }
```

ST specifies single section from the standard built-in tables.
RA specifies single angle with reverse Y-Z axes (see Section 2.5.2).
D specifies double channel.
LD specifies long leg, back to back, double angle.
SD specifies short leg, back to back, double angle.
T specifies tee section cut from I shape beams.
CM specifies composite section, available with I shape beams.
TC specifies beams with top cover plate.
BC specifies beams with bottom cover plate.
TB specifies beams with top and bottom cover plates.

```
table-name = Table section name like W8X18, C15X33 etc.
```

```
* \{ SP  f1  
    WP  f2  
    TH  f3  
    WT  f4  
\}
```

**additional-spec =**

```
\{ DT  f5  
  OD  f6  
  ID  f7  
  CT  f8  
  FC  f9  
\}
```

**SP**  
`f1=` his set describes the spacing (f1) between angles or channels if double angles or double channels are used. f1 defaults to 0.0 if not given.
WP   f2= Width (f2) of the cover plate if a cover plate is used with W, M, S, or HP sections.
TH   f3= Thickness (f3) of plates or tubes.
WT   f4= Width (f4) of tubes, where TUBE is the table-name.
DT   f5= Depth (f5) of tubes.
OD   f6= Outside diameter (f6) of pipes, where PIPE is the table-name.
ID   f7= Inside diameter (f7) of pipes.
CT   f8= Concrete thickness (f8) for composite sections.
FC   f9= Compressive strength (f9) of the concrete for composite sections.

Notes
All values f1-9 must be supplied in current units.
Some important points to note in the case of the composite section are:

1) The width of the concrete slab is assumed to be the width of the top flange of the steel section + 16 times the thickness of the slab.
2) In order to calculate the section properties of the cross-section, the modular ratio is calculated assuming that:

\[ E_s = \text{Modulus of elasticity of steel} = 29000 \text{ Ks}i. \]
\[ E_c = \text{Modulus of elasticity of concrete} = 1802.5 \text{ Ks}i \]
where FC (in Ksi) is defined earlier.

Prismatic Property Specification

Purpose
The following commands are used to specify section properties for prismatic cross-sections.

General format:
For the PRISMATIC specification, properties are provided directly as follows:

\[
\begin{align*}
&\left\{ \begin{array}{l}
AX \quad f1 \\
IX \quad f2 \\
IY \quad f3 \\
IZ \quad f4 \\
AY \quad f5 \\
AZ \quad f6 \\
YD \quad f7 \\
ZD \quad f8 \\
YB \quad f9 \\
ZB \quad f10 
\end{array} \right. \\
\text{property-spec.} = \\
* \end{align*}
\]

AX   f1 = Cross sectional area of the member. If omitted, the area is calculated from the YD and ZD dimensions.
IX   f2 = Torsional constant.
IY  \( f_3 \) = Moment of inertia about local y-axis.
IZ  \( f_4 \) = Moment of inertia about local z-axis (usually major).
AY  \( f_5 \) = Effective shear area in local y-axis.
AZ  \( f_6 \) = Effective shear area in local z-axis.
YD  \( f_7 \) = Depth of the member in local y direction. (Diameter of section for circular members)
ZD  \( f_8 \) = Depth of the member in local z direction.
YB  \( f_9 \) = Depth of stem for T-section.
ZB  \( f_{10} \) = Width of stem for T-section or bottom width for TRAPEZOIDAL section.

**Examples of Member Property Specification**

This section illustrates the various options available for MEMBER PROPERTY specification

**Example**

UNIT INCHES
MEMBER PROPERTIES
1 TO 5 TABLE ST W8X31
9 10 TABLE LD L40304 SP 0.25
12 TO 15 PRISMATIC AX 10.0 IZ 1520.0 IY 600.
17 18 TA ST PIPE OD 2.5 ID 1.75
20 TO 25 TA ST TUBE DT 12. WT 8. TH 0.5
27 29 32 TO 40 -
42 PR AX 5. IZ 400. IY 33. IX 0.2 YD 9. ZD 3.
56 TA TC W12X26 WP 4.0 TH 0.3
57 TA CM W14X34 CT 5.0 FC 3.0

This example shows each type of member property input. Members 1 to 5 are wide flanges selected from the AISC tables; 9 and 10 are double angles selected from the AISC tables; 12 to 15 are prismatic members with no shear deformation; 17 and 18 are pipe sections; 20 to 25 are tube sections; 27, 29, 32 to 40, and 42 are prismatic members with shear deformation; Member 56 is a wideflange W12X26 with a 4.0 in. wide cover plate of thickness 0.3 inches at the top. Member 57 is a composite section with a concrete slab thickness of 5.0 inches at the top of a wide flange W14X34. The compressive strength of the concrete in the slab is 3.0 ksi.

**8- Member Release Specification**

**Purpose**
This set of commands may be used to release specified degrees of freedoms at the ends of frame members.

**General format:**
MEMBER RELEASES

\[
\begin{align*}
\text{member-list} \left\{ \begin{array}{c}
\text{START} \\
\text{END}
\end{array} \right\} \rightarrow \begin{array}{c}
\text{FX} \\
\text{FY} \\
\text{FZ} \\
\text{MX} \\
\text{MY} \\
\text{MZ}
\end{array}
\end{align*}
\]

Where FX through MZ represent force-x through moment-z in the member local axes.

Example
MEMBER RELEASE
1 3 TO 9 11 12 START MY MZ
1 10 11 13 TO 18 END MZ

In the above example, local moment y and moment z are released for the first set of members at their start joints (as specified in MEMBER INCIDENCES), while only moment z is released for the second set of members at their end joints. Note that members 1 and 11 are released at both start and end joints.

9- Member Truss Specification

Purpose
This command may be used to model a specified set of members as TRUSS members.

Description
This specification may be used to specify TRUSS type members in a PLANE, SPACE or FLOOR structure. The TRUSS members are capable of carrying only axial forces. Typically, bracing members in a PLANE or SPACE frame will be of this nature.

General format:
MEMBER TRUSS
member-list

Note that this command is superfluous when a TRUSS type structure has already been specified.

Example
MEMB TRUSS
1 TO 8 10 12 14 15

Notes
The TRUSS member has only one degree of freedom-the axial deformation. It is not equivalent to a frame member with moment releases at both ends.
10- Constant Specification

Purpose
This command may be used to specify the material properties (Modulus of Elasticity, Poisson's ratio, Density and Co-efficient of linear expansion) of the members and elements.

General format:
CONSTANTS
\[
\begin{align*}
\text{E} & \quad \text{POISSON} & \quad \text{DENSI}T\text{Y} & \quad \text{ALPHA} & \quad \text{MEMBER member list} \\
\text{f1} & & & & \quad \text{ALL}
\end{align*}
\]

E specifies Young's Modulus. This value must be provided as the first item in the Constants list.

POISSON specifies Poisson's Ratio. This value is used for calculating the Shear Modulus \( G = 0.5xE/(1+\text{POISSON}) \).

DENSITY specifies weight density.

ALPHA Co-efficient of thermal expansion.

f1 Value of the corresponding constants. For E, POISSON and DENSITY, material names can be provided instead of f1. Appropriate values will be automatically assigned. The PRINT MATERIAL PROPERTY command of material CONSTANTS. Current list of material names consist of STEEL, CONCRETE & ALUMINUM.

Example
CONSTANTS
E 29000.0 ALL
DENSITY STEEL MEMB 14 TO 29

Notes
1) The value for E must always be given first in the Constants list.
2) All numerical values must be provided in the current units system.

11- Global Support Specification

Purpose
This set of commands may be used to specify the SUPPORT conditions for support parallel to the global axes.

General format:
SUPPORTS
\[
\begin{align*}
\text{PINNED} & \quad \text{FIXED} & \quad \text{release-spec. [ spring-spec. ]}
\end{align*}
\]

joint-list
**Description**

PINNED support is a support which has translational, but no rotational restraints. In other words, the support has no moment carrying capacity. A FIXED support has both translational and rotational restraints. A FIXED support can be released in the global directions as described in release-spec (FX for force-X through MZ for moment-Z). Also, a fixed support can have spring constants as described in spring-spec (translational spring in global X-axis as KFX through rotational spring in global Z-axis as KMZ). Corresponding spring constants are f1 through f6. Note that the rotational spring constants are always per degree of rotation. No more than five releases may be provided. If both release specifications and spring specifications are to be supplied for the same support joint, release specifications must come first.

**Example**

SUPPORTS
1 TO 4  7 PINNED
5  6 FIXED BUT FX MZ
8  9 FIXED BUT MZ KFX 50.0 KFY 75.
18 21 FIXED
27 FIXED BUT KFY 125.0

In this example, joints 1 to 4 and joint 7 are pinned, no moments are carried by those supports. Joints 5 and 6 are fixed in all directions except in force-X and moment-Z, force-X and moment-Z of these supports are released. Joints 8 and 9 are fixed with a release in moment-Z and have springs in the global X and Y directions with corresponding spring constants of 50 and 75 units respectively. Joints 18 and 21 are fixed in all translational and rotational directions. Joint 27 is fixed in all directions with 125 unit spring in the global Y direction.

**12- Draw Specifications**

**Purpose**

This set of commands may be used to generate printer plots of structure geometry and results as parts of the output.

**Description**

Besides interactive graphics, STAAD-III has features to provide commands to plot structural geometry, analysis results etc. as part of the STAAD-III output file.
Note that, these output files (.ANL files) should be printed only through the PRINT option of STAAD-UTIL menu. Plots can also be displayed by the STAAD-VIEW option of STAAD-UTIL.

Plots are of very high-resolution and practically all the 8/9/24 pin dot matrix and laser printers are supported.

DRAW command is used to create all the plots in the output. The following is the format of DRAW commands.

\[
\text{DRAW} \begin{cases}
\text{ISOMETRIC} \\
\text{JOINT} \\
\text{MEMBER} \\
\text{SUPPORT} \\
\text{PROPERTY} \\
\text{SHAPE} \\
\text{LOAD} \quad \text{ln} \\
\text{DFDRAW} \quad \text{ln} \\
\text{SCDRAW} \quad \text{ln} \\
\text{MSDRAW} \quad \text{ln} \quad \text{force-spec} \\
\text{BMDRAW} \quad \text{ln} \quad \text{force-spec} \\
\text{VALUE}
\end{cases} \quad \text{(LIST list – spec.)}
\]

\[\text{ln} \quad = \quad \text{Load number to be considered}\]

\[\text{force-spec} = \begin{cases}
\text{FX} \\
\text{FY} \\
\text{MZ} \\
\text{FZ} \\
\text{MY}
\end{cases}\]

**Notes**

1) Following commands may be used anywhere in the input.

ISOMETRIC = Draw isometric view.
JOINT = Display joint numbers.
MEMBER = Display member numbers.
SUPPORT = Display support icons.
PROPERTY = Display property names.
LOAD = Display load icons. Obviously, this command can be used only after the loadings are provided.

2) Following commands are related to results and should be used only after the PERFORM ANALYSIS command.

DFDRAW = Draw deflected shape.
MODRAW = Draw mode shape.
SCDRAW = Draw section displacement.
MSDRAW = Display force/moment diagram on the entire structure for specified in (load number).

BMDRAW = Display force/moment diagram for independent members as listed in LIST. No more than 2 member lists are allowed. Use multiple DRAW commands to display force/moment diagrams for independent members.

Example

DRAW ISOMETRIC MEMBER SUPPORT PROPERTY
DRAW SHAPE SUPPORT
DRAW ISOMET MSDRAW 2 MZ VALUE

13- Loading Specifications

Purpose
This section describes the various loading options available in STAAD-III. The following command may be used to initiate a new load case.

General format:
LOADING i1 (any load title)

i1 = any unique integer number (upto four digits) to identify the load case. This number need not be sequential with previous load number.

The LOADING command initiates a new load case. Under this heading, all different loads related to this loading number can be input. These different kinds of loads are described below.

Selfweight Load Specification

Purpose
This command may be used to calculate and apply the SELFWEIGHT of the structure for analysis.

General format:
SELFWEIGHT \( \begin{bmatrix} X \\ Y \\ Z \end{bmatrix} \) f1

This command is used if the self-weight of the structure is to be considered. The self-weight of every active member is calculated and applied as a uniformly distributed member load.

X, Y, & Z represent the global direction in which the selfweight acts.
f1 = The factor to be used to multiply the selfweight.

This command may also be used without any direction and factor specification. Thus, if specified as "SELFWEIGHT", loads will be applied in the negative global Y direction with a factor of unity.

**Notes**
Density must be provided for calculation of the self weight.

**Joint Load Specification**

**Purpose**
This set of commands may be used to specify JOINT loads on the structure.

**General format:**

```
JOINT LOAD
```

```
* FX f1
FY f2
FZ f3
MX f4
MY f5
MZ f6
```

FX, FY and FZ specify a force in the corresponding global direction.
MX, MY and MZ specify a moment in the corresponding global direction.
f1, f2 ... f6 are the values of the loads.

**Example**

```
JOINT LOAD
3 TO 7 9 11 FY -17.2 MZ 180.0
5 8 FX 15.1
12 MX 180.0 FZ 6.3
```

**Notes**
Joint numbers may be repeated where loads are meant to be additive in the joint.

**Member Load Specification**

**Purpose**
This set of commands may be used to specify MEMBER loads on frame members.

**General format:**

```
MEMBER LOAD
```

```
member-list
   { UNI or U MOM direction-spec f1, f2, f3, f4
CON or CMOM direction-spec f5, f6, f4
LIN local-spec f7, f8, f9
TRAP direction-spec f10, f11, f12, f13
```
direction-spec = \[
\begin{pmatrix}
X \\
Y \\
Z \\
GX \\
GY \\
GZ \\
PX \\
PY \\
PZ
\end{pmatrix}
\]
local-spec = \[
\begin{pmatrix}
X \\
Y \\
Z
\end{pmatrix}
\]

UNI or UMOM specifies a uniformly distributed load or moment with a value of \(f_1\), at a distance of \(f_2\) from the start of the member to the start of the load, and a distance of \(f_3\) from the start of the member to the end of the load. The load is assumed to cover the full member length if \(f_2\) and \(f_3\) are omitted.

CON or CMOM specifies a concentrated force or moment with a value of \(f_5\) applied at a distance of \(f_6\) from the start of the member. \(f_6\) will default to half the member length if omitted.

\(f_4 = \) Perpendicular distance from the member shear center to the plane of loading. The value is positive in the general direction of the parallel (or close to parallel) local axis.

LIN specifies a linearly decreasing or increasing, or a triangular load. If the load is linearly increasing or decreasing then \(f_7\) is the value at the start of the member and \(f_8\) is the value at the end. If the load is triangular, then \(f_7\) and \(f_8\) are input as zero and \(f_9\) is the value of the load in the middle of the member.

TRAP specifies a trapezoidal linearly varying load which may act over the full or partial length of a member and in a local, global or projected direction. The starting load value is given by \(f_{10}\) and the ending load value by \(f_{11}\). The loading location is given by \(f_{12}\), the loading starting point, and \(f_{13}\), the stopping point. Both are measured from the start of the member. If \(f_{12}\) and \(f_{13}\) are not given, the load is assumed to cover the full member length.

\(X, Y, \) & \(Z\) in the direction-spec and local-spec specify the direction of the load in the local (member) \(x, y\) and \(z\)-axes. \(GX, GY, \) & \(GZ\) in the direction-spec specify the direction of the load in the global \(X, Y,\) and \(Z\)-axes. \(PX, PY\) and \(PZ\) may be used if the load is to be along the projected length of the member in the corresponding global direction.

Notes
Members may be repeated where the loads in the member are meant to be additive. Also, note that specifying global axes is not permissible for the linear load.
Temperature Load Specification for Members

Purpose
This command may be used to specify TEMPERATURE loads or strain loads on members and elements.

General format:

**TEMPERATURE LOAD**
member-list **TEMP** f1 f2

f1 = The change in temperature which will cause axial elongation in the members or uniform volume expansion in elements. The temperature unit is the same as the unit chosen for the coefficient of thermal expansion ALPHA under the CONSTANT command.

f2 = The temperature differential from the top to the bottom of the member (T_top surface - T_bottom surface). If f2 is omitted, no bending will be considered.

Example

TEMP LOAD
1 TO 9 15 17 TEMP 70.0
18 TO 23 TEMP 90.0 66.0

Notes
Members may be repeated where the loads in the member are meant to be additive.

Support Displacement Load Specification

Purpose
This command may be used to specify SUPPORT DISPLACEMENT load on supports of the structure.

General format:

**SUPPORT DISPLACEMENT** (LOAD)

support-joint-list \[
\begin{align*}
FX \\
FY \\
FZ \\
MX \\
MY \\
MZ
\end{align*}
\] f1

With this command, the support displacement is modeled as a load. Note that displacement cannot be specified in a direction in which the support is released.

FX, FY, FZ specify translational displacements in global X, Y, and Z directions respectively. MX, MY, MZ specify rotational displacements in global X, Y, and Z directions.
f1 = Value of the corresponding displacement. For translational displacements, the unit is in the currently specified length unit, while for rotational displacements the unit is always in degrees.

**Example**

UNIT INCHES
SUPPORT DISPL
5 TO 11 13 FY -0.25
19 21 TO 25 MX 15.0

In this example, the first support list will be displaced by 0.25 inch in the negative global Y direction, while the second support list will be rotated by 15 degrees around global X-axis.

**Notes**
Load may be repeated where the effects are meant to be additive.

---

### 14- Load Combination Specification

**Purpose**
This command may be used to combine the results of the analysis. The combination may be algebraic, SRSS and a combination of both.

**General format:**

```
LOAD COMBINATION  i  a1
```

- \(i\) = Load combination number (any integer value less than 150 but not the same as any previously defined primary load case number).
- \(a1\) = Any title for the load combination.
- \(i1, i2, ...\) represents the load case numbers which are to be combined.
- \(f1, f2, ...\) represents corresponding factors to be applied to loadings.

**Example**
Several combination examples are provided to illustrate the possible combination schemes -

```
LOAD COMBINATION 7 DL+LL+WL
1 0.75 2 0.75 3 1.33
```
The item above (LOAD COMBINATION 7) illustrates a simple algebraic combination.

**Notes**
This option combines the results of the analysis in the specified manner. It does not analyze the structure for the combined loading.
15- Analysis Specification

Purpose
STAAD-III analysis options include linear static analysis, P-Delta (or second order analysis), Nonlinear analysis, and several types of Dynamic analysis. This command is used to specify the analysis request. In addition, this command may be used to request various analysis related data like load info, statics check info, mode shapes etc.

General format:

\[
\begin{cases}
\text{PERFORM} \\
\text{PDELTA (n)} \\
\text{NONLINEAR (n)}
\end{cases}
\]

ANALYSIS

Where n = no. of iterations desired (default value of n = 1).

This command directs the program to perform the analysis which includes:

a) checking whether all information is provided for the analysis;
b) forming the joint stiffness matrix;
c) checking the stability of the structure;
d) solving simultaneous equations, and
e) computing the member forces and displacements.
f) If P-Delta analysis is specified, forces and displacements are recalculated, taking into consideration the P-Delta effect.
g) Non-linear analysis will take the geometric non-lineality as well the P-Delta effects into account.
h) in each of the "n" iterations of the PDELTA analysis, the load vector will be modified to include the secondary effect generated by the displacements caused by the previous analysis.

Without one of these commands, no analysis will be performed. These ANALYSIS commands can be repeated if multiple analyses are needed at different phases.

Notes
STAAD-III allows multiple analyses in the same run. Multiple analysis may be used for the following purposes:

1) Successive analysis and design cycles in the same run result in optimized design. STAAD-III's live relational database automatically updates changes in member cross-sectional sizes. Thus the entire process is automated.

Refer to Example 1 in the Getting Started & Examples manual for detailed illustration.

2) Multiple analysis may be used for load-dependent structures. For example, structures with bracing members are analyzed in several steps. The bracing members are assumed to
take Tension load only. Thus, they need to be activated and inactivated based on the direction of lateral loading.

The entire process can be modeled in one STAAD-III run using multiple PERFORM ANALYSIS commands. The STAAD-III run database automatically keeps track of results for different runs and is capable of generating a design based on load combinations provided.

3) The user may also use Multiple Analysis to model change in other characteristics like SUPPORTS, RELEASES, SECTION PROPERTIES etc.

### 16- Load List Specification

**Purpose**
This command allows specification of a set of active load cases. All load cases made active by this command remain active until a new load list is specified.

**General format:**

```
LOAD LIST \{load-list \}
```

**Description**
This command is used to activate the load cases listed in this command and, in a sense, deactivate all other load cases not listed in this command. In other words, the loads listed are used for printing output and in design for performing the specified calculations. Note that, when PERFORM ANALYSIS command is used, the program internally uses all load cases, regardless of LOAD LIST command.

**Example**

```
LOAD LIST ALL
PRINT MEMBER FORCES
LOAD LIST 1 3
PRINT SUPPORT REACTIONS
```

In this example, member forces will be printed for all load cases, whereas loading 1 and 3 will be used for printing support reactions.

**Notes**
The LOAD LIST command may be used for multiple analysis situations when an analysis needs to be performed with a selected set of load cases only.
17- Section Specification

Purpose
This command is used to specify sections along the length of frame member for which forces and moments are required.

General format:

\[
\text{SECTION } f_1, f_2 \ldots f_5 \quad \left\{ \text{MEMBER Memb-list} \right\} \\
\left( \text{ALL} \right)
\]

Description
This command specifies the sections, in terms of fractional member lengths, at which the forces and moments are considered for further processing.

\( f_1, f_2 \ldots f_5 \) = Section (in terms of the fraction of the member length) provided for the members. Maximum number of sections is 5, including one at the start and one at the end. In other words, no more than three intermediate sections are permissible per SECTION command.

Example

\[
\text{SECTION 0.0 0.5 1.0 MEMB 1 2} \\
\text{SECTION 0.25 0.75 MEMB 3 TO 7} \\
\text{SECTION 0.6 MEMB 8}
\]

In this example, first the members 1 and 2 are set to section values of 0.0, 0.5, and 1.0, i.e. at the start, mid point and end. The members 3 to 7 are specified by the next SECTION command where sections 0.25 and 0.75 are set.

In the next SECTION command, member 8 has its section specified at 0.6. The remainder of the members will have no effect since no intermediate SECTION is provided for them. If no section value is given for any member, it defaults to 0.0 and 1.0 (i.e. start and end). For example, the start and end forces of the members will be used for design, if no section is specified. As mentioned earlier, no more than three intermediate sections are allowed per SECTION command. However, if more than three intermediate sections are involved, they can be examined by repeating the SECTION command after completing the required calculations. The following example will clarify.

Example

\[
\text{SECTION 0.2 0.4 0.5 ALL} \\
\text{PRINT SECTION FORCES} \\
\text{SECTION 0.6 0.75 0.9 ALL}
\]
PRINT SECTION FORCES

In this example, first forces at 3 intermediate sections (namely 0.2, 0.4 and 0.5) are printed and then forces at an additional 3 sections (namely 0.6, 0.75 and 0.9) are printed. This gives the user the section forces at more than three intermediate sections.

Notes
1) The SECTION command just specifies the sections. Use the PRINT SECTION FORCES command after this command to print out the forces and moments at the specified sections.
2) This is a secondary analysis command. Note that the analysis must be performed before this command may be used.

18- Print Specifications
Purpose
This command is used to direct the program to print various modeling information and analysis results. STAAD-III offers a number of versatile print commands that can be used to customize the output.

General format for data related print commands:

```
PRINT \{ JOINT COORDINATES
       MEMBER INFORMATION
       MEMBER PROPERTIES
       PRINT MATERIAL PROPERTIES
       SUPPORT INFORMATION
       ALL
       CG \}

\{ (ALL)
   LIST list of items i.e. joint, or member \}
```

General format to print analytical results:

```
PRINT \{ (JOINT) DISPLACEMENTS
       (MEMBER) FORCES
       (SUPPORT) REACTIONS
       ANALYSIS RESULTS
       (MEMBER) SECTION FORCES
       (MEMBER) STRESSES
       \}

\{ (ALL)
   LIST list of items i.e. joint, or member \}
```

General format to print entire steel table:

```
PRINT ENTIRE (TABLE)
```

Description
Note that the list of items are not applicable for PRINT ANALYSIS RESULTS, and PRINT SUPPORT REACTIONS command.
PRINT JOINT COORDINATES command prints all interpreted coordinates of joints.
PRINT MEMBER INFORMATION command prints all member information, including member length, member incidences, beta angles, whether or not a member is a truss member and the member release conditions at start and end of the member (1=released, 0 = not released).

PRINT MEMBER PROPERTIES command prints all member properties including cross sectional area, moments of inertia, and section moduli in both axes. Units for the properties are always INCH or CM (depending on FPS or METRIC) regardless of the unit specified in UNIT command.

PRINT MATERIAL PROPERTIES command prints all material properties for the members, including E (modulus of elasticity), G (shear modulus), weight density and coefficient of thermal expansion (alpha) for frame members.

PRINT SUPPORT INFORMATION command prints all support information regarding their fixity, releases and spring constant values, if any.

PRINT ALL command is equivalent to last five print commands combined. This command prints joint coordinates, member information, member properties, material properties and support information, in that order.

PRINT CG command prints out the coordinates of the center of gravity of the structure.

PRINT (JOINT) DISPLACEMENTS command prints joint displacements in a tabulated form. The displacements for all six directions will be printed for all specified load cases. The length unit for the displacements is always INCH or CM (depending on FPS or METRIC unit) regardless of the unit specified in UNIT command.

PRINT (MEMBER) FORCES command prints member forces (i.e. axial, shear-y, shear-z, torsion bending-y and bending-z) in a tabulated form by member, for all specified load cases.

PRINT (SUPPORT) REACTIONS command prints support reactions in a tabulated form, by support, for all specified load cases.

PRINT ANALYSIS RESULTS command is equivalent to the last three commands combined. With this command, the joint displacements, support reactions and member forces, in that order, are printed.

PRINT (MEMBER) SECTION FORCES command prints member forces at the intermediate sections specified with a previously input SECTION command. The printing is done in a tabulated form, by member, for all specified load cases.

PRINT (MEMBER) STRESSES command tabulates member stresses at the start joint, end joint and all specified intermediate sections. These stresses include axial (i.e. axial force over the area), bending-y (i.e. moment-y over section modulus in local y-axis),
bending-z (i.e. moment-z over section modulus in local z-axis), shear stresses in both local y and z directions and combined (absolute combination of axial, bending-y and bending-z) stresses. For PRISMATIC sections, if AY and/or AZ is not provided, the full cross-sectional area (AX) will be considered in shear stress calculations. If the JOINT option is used, forces and moments at the nodal points are also printed out in addition to the centroid of the element.

**Example**

PERFORM ANALYSIS
PRINT SUPPORT REACTIONS
PRINT JOINT DISPLACEMENTS LIST 1 TO 50
PRINT MEMBER FORCES LIST 101 TO 124

**Notes**

1) The output generated by these commands are based on the current unit system. The user may wish to verify the current unit system and change it if necessary.

2) Results may be printed for all joints/members or based on a specified list.

---

**19- Steel Design Specifications**

This section describes all the specifications necessary for structural steel design.

**Parameter Specifications**

**Purpose**

This set of commands may be used to specify the parameters required for steel design.

**General format:**

PARAMETER

```

PARAMETER

\( \left\{ \begin{array}{c}
\text{AASHTO} \\
\text{AISC} \\
\text{AUSTRALIAN} \\
\text{BRITISH} \\
\text{CANADIAN} \\
\text{FRENCH} \\
\text{GERMAN} \\
\text{INDIA} \\
\text{JAPAN} \\
\text{LRFD} \\
\text{NORWAY} \\
\end{array} \right\}
```

Parameter - name \( f1 \) \( \left\{ \begin{array}{c}
\text{MEMBER member-list} \\
\text{ALL} \\
\end{array} \right\}

**Description**

parameter-name - refers to the "PARAMETER NAME" (s) listed in the parameter table (1) contained in the Steel Design section. For AISC Allowable Stress Design - see Table 3.1 and AASHTO based design. For AISC LRFD Design - see Table 3.2 For steel design per other codes, refer to the relevant section.
f1 = Value of the parameter.
The user can control the design through specification of proper parameter.

Notes
1) All unit sensitive values should be in the current unit system. 
2) For default values of the parameters, refer to table (1).

### TABLE (1) : AMERICAN STEEL DESIGN PARAMETERS

<table>
<thead>
<tr>
<th>Parameter name</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>KY</td>
<td>1.0</td>
<td>K value in local y-axis, Usually, this minor axis</td>
</tr>
<tr>
<td>KZ</td>
<td>1.0</td>
<td>K value in local z-axis, Usually, this major axis</td>
</tr>
<tr>
<td>LY</td>
<td>Member Length</td>
<td>Length in local y-axis tu calculate slenderness ratio.</td>
</tr>
<tr>
<td>LZ</td>
<td>Member Length</td>
<td>Same as above except in z-axis (major).</td>
</tr>
<tr>
<td>FYLD</td>
<td>=36 KSI =248 MPa</td>
<td>Yield strength of steel in current units (Fy).</td>
</tr>
<tr>
<td>NSF</td>
<td>1.0</td>
<td>Net section factor for tension members.</td>
</tr>
<tr>
<td>UNL</td>
<td>Member Length</td>
<td>Unsupported length for calculating allowable bending stress.</td>
</tr>
<tr>
<td>UNF</td>
<td>1.0</td>
<td>Same as above provided as a fraction of actual member length</td>
</tr>
<tr>
<td>CB</td>
<td>1.0</td>
<td>Cb value as used in section 1.5 of AISC 0.0 = Cb value to be calculated. Any other value will mea the value to be used in design.</td>
</tr>
<tr>
<td>SSY</td>
<td>0.0</td>
<td>0.0= Sidesway in local y-axis. 1.0= No sidesway.</td>
</tr>
<tr>
<td>SSZ</td>
<td>0.0</td>
<td>Same as above except I local z-axis.</td>
</tr>
<tr>
<td>CMY</td>
<td>0.85 for sidesway and calculated for no sidesway</td>
<td>Cm value in local y and z axes.</td>
</tr>
<tr>
<td>CMZ</td>
<td>0.0</td>
<td></td>
</tr>
<tr>
<td>MAIN</td>
<td>0.0</td>
<td>0.0 = check for selenderness. 1.0 = suppress selenderness check.</td>
</tr>
<tr>
<td>STIFF</td>
<td>Member Length</td>
<td>Spacing of stiffeners for plate girder design.</td>
</tr>
<tr>
<td>TRACK</td>
<td>0.0</td>
<td>0.0 = Suppress critical member stresses 1.0 = Print all critical member stresses. 2.0 = Print expanded output.</td>
</tr>
<tr>
<td>DMAX</td>
<td>45 in= 1143 mm</td>
<td>Maximum allowable depth.</td>
</tr>
<tr>
<td>DMIN</td>
<td>0.0</td>
<td>Minimum allowable depth.</td>
</tr>
<tr>
<td>RATIO</td>
<td>1.0</td>
<td>Permissible ratio of the actual to allowable stresses.</td>
</tr>
<tr>
<td>BEAM</td>
<td>0.0</td>
<td>0.0 = design only for end moments for those at locations specified by SECTION command. 1.0 = calculate moments at tenth points along the beam, and use the maximum, Mz location for design.</td>
</tr>
</tbody>
</table>
Example
PARAMETERS
CODE AISC
KY 1.5 MEMB 3 7 TO 11
NSF 0.75 ALL
RATIO 0.9 ALL

Code Checking Specification

Purpose
This command may be used to perform the CODE CHECKING operation.

General format:

\[
\text{CHECK \ CODE} \begin{cases} 
\text{MEMBER member-list} \\
\text{ALL} 
\end{cases}
\]

Description
This command checks the specified members against the desired code. The results of the code checking are tabulated by each member. Refer to Section 3 of this manual for detailed information.

Notes
The output of this command may be controlled using the TRACK parameter. Three levels of details are available. Refer to the appropriate Steel Design section for more information on the TRACK parameter.

Member Selection Specification

Purpose
This command may be used to perform the MEMBER SELECTION operation.

General format:

\[
\text{SELECT} \begin{cases} 
\text{MEMBER member-list} \\
\text{ALL} 
\end{cases}
\]

Description
By this command, the program selects specified members based on the parameter value restrictions and specified code. The selection is done using the last results from analysis and iterating on sections until a least weight size is obtained.

Notes
The output of this command may be controlled using the TRACK parameter. Three levels of details are available. Refer to the appropriate Steel Design section for more information on the TRACK parameter.

**Member Selection by Optimization**

**Purpose**
This command performs member selection using an optimized technique based on multiple analysis/design iterations.

**General format:**

```
SELECT OPTIMIZED
```

**Description**
By this command, the program selects all members based on a state-of-the-art optimization technique. This method requires multiple stiffness matrix analyses as well as iteration of sizes until an overall structure least weight is obtained. This command should be used with caution, since it will require longer computer time to solve a structure.

**Notes**
1) The output of this command may be controlled using the TRACK parameter. Three levels of details are available. Refer to the appropriate Steel Design section for more information on the TRACK parameter.

2) This command may require multiple iterations involving analysis/design cycles and therefore may be time consuming.

**Steel Take Off Specification**

**Purpose**
This command may be used to obtain a summary of all steel sections being used along with their lengths and weights.

**General format:**

```
STEEL (MEMBER) TAKE (OFF)
```

**Description**
This command provides a complete listing of all different steel table sections used in the structure. The tabulated listing will include total length of each section name and its total weight. This can be helpful in estimating steel quantities.

The MEMBER option list each member length and weight by number, profile-type, length and weight.

**Notes**
This facility may be very effectively utilized to obtain a quick estimate of the structural steel quantity.
20- Concrete Design Specifications
This section describes the specifications for concrete design. The concrete design procedure implemented in STAAD-III consists of the following steps:
1) Initiating the design.
2) Specifying of parameters.
3) Specifying design requirements.
4) Requesting quantity take-off.
5) Terminating the design

Design Initiation
Purpose
This command is used to initiate the concrete design.
General format:

START CONCRETE DESIGN

Description
This command initiates the concrete design specification. With this, the design parameters are automatically set to the default values (as shown on table 2). Without this command, none of the following concrete design commands will be recognized.

Notes
This command must be present before any concrete design command is used.

Concrete Design-Parameter Specification
Purpose
This set of commands may be used to specify parameters to control the concrete design.
General format:

CODE
\begin{align*}
\text{ACI} \\
\text{BRITISH} \\
\text{CANADIAN} \\
\text{FRENCH} \\
\text{GERMAN} \\
\text{INDIA} \\
\text{JAPAN} \\
\text{NORWAY}
\end{align*}

\text{parameter-name f1} \begin{cases}
\text{MEMBER member list} \\
(\text{ALL})
\end{cases}

Description
Parameter-name refers to the concrete parameters described in table f1 = is the value of the parameter. Note that this value is always input in current units. The UNIT command is also accepted during any phase of concrete design.

Notes
1) All parameter values are provided in the current unit system.
2) For default values of parameters, refer to table (2)

### TABLE (2): AMERICAN CONCRETE DESIGN PARAMETERS

<table>
<thead>
<tr>
<th>Parameter name</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>FYMAIN</td>
<td>60000 psi = 414 MPa</td>
<td>Yield stress for main reinforcing steel (fy)</td>
</tr>
<tr>
<td>FYSEC</td>
<td>60000 psi = 414 MPa</td>
<td>Yield stress for secondary reinforcing steel (fy)</td>
</tr>
<tr>
<td>FC</td>
<td>4000 psi = 27.5 MPa</td>
<td>Compressive strength of concrete (f’c)</td>
</tr>
<tr>
<td>CLT</td>
<td>1.5 in = 38 mm</td>
<td>Clear cover for top reinforcement.</td>
</tr>
<tr>
<td>CLB</td>
<td>1.5 in = 38 mm</td>
<td>Clear cover for bottom reinforcement.</td>
</tr>
<tr>
<td>CLS</td>
<td>1.5 in = 38 mm</td>
<td>Clear cover for side reinforcement.</td>
</tr>
<tr>
<td>MINMAIN</td>
<td>Number 4 bar</td>
<td>Minimum main reinforcement bar size (# 4 – # 18)</td>
</tr>
<tr>
<td>MINSEC</td>
<td>Number 4 bar</td>
<td>Minimum secondary reinforcement bar size.</td>
</tr>
<tr>
<td>MAXMAIN</td>
<td>Number 18 bar</td>
<td>Maximum main reinforcement bar size.</td>
</tr>
<tr>
<td>SFACE</td>
<td>0.0</td>
<td>Face of support location at start of beam.</td>
</tr>
<tr>
<td>EFACE</td>
<td>0.0</td>
<td>Face of support location at end of beam (Note: Both SFACE &amp; EFACE are input as positive members)</td>
</tr>
<tr>
<td>REINF</td>
<td>0.0</td>
<td>Tied column. A value of 1.0 will mean spiral.</td>
</tr>
<tr>
<td>MMAG</td>
<td>1.0</td>
<td>A factor by which the design moment will be magnified.</td>
</tr>
<tr>
<td>WIDTH</td>
<td>ZD</td>
<td>Width of concrete member. This value defaults to ZD as provided under MEMBER PROPERTIES.</td>
</tr>
<tr>
<td>DEPTH</td>
<td>YD</td>
<td>Depth of concrete member. This value defaults to YD as provided under MEMBER PROPERTIES.</td>
</tr>
<tr>
<td>NSECTION</td>
<td>10</td>
<td>Number of equally spaced sections to be considered in finding critical moments for beam design.</td>
</tr>
</tbody>
</table>
| TRACK         | 0.0 | BEAM DESIGN:
0.0 = Critical moments will not be printed out with beam design report.
1.0 = Critical moment will be printed out.
2.0 = print out required steel areas for all intermediate sections specified by NSECTION.

COLUMNS DESIGN:
0.0 = prints out detailed design results.
1.0 = prints out column interaction analysis results in addition to TRACK=0.0 output.
2.0 = prints out a schematic interaction diagram and intermediate interaction values in addition to all of above. |
Concrete Design Command

Purpose
This command may be used to specify the type of design required. Members may be
designed as BEAM, or COLUMN.

General format:

DESIGN { BEAM COLUMN } member-list

Description
Members to be designed must be specified as BEAM, or COLUMN. Note that members,
once designed as beam, cannot be redesigned as a column again, or vice versa.

Concrete Take Off Command

Purpose
This command may be used to obtain an estimate of the total volume of the concrete,
reinforcement bars used and their respective weights.

General Format:

CONCRETE TAKE OFF

Description
This command can be issued to print the total volume of concrete and the bar numbers
and their respective weight for the members designed.

SAMPLE OUTPUT:

************** CONCRETE TAKE OFF **************
(FOR BEAMS AND COLUMNS DESIGNED ABOVE)

TOTAL VOLUME OF CONCRETE = 87.50 CU.FT

BAR SIZE   WEIGHT
NUMBER     (in lbs)
-----------
4          805.03
6          91.60
8          1137.60
9          653.84
11         818.67
-----------
*** TOTAL = 3506.74

Notes
This command may be used very effectively for quick quantity estimates.

Concrete Design Terminator

Purpose
This command must be used to terminate the concrete design.

General format:

END  CONCRETE  DESIGN

Description
This command terminates the concrete design, after which normal STAAD-III commands resume.

Example

START CONCRETE DESIGN
CODE ACI
FYMAIN 40.0 ALL
FC 3.0 ALL
DESIGN BEAM 1 TO 4 7
DESIGN COLUMN 9 12 TO 16
END

21- Footing Design Specifications

Purpose
This set of commands may be used to specify footing design requirements. describes the process of design initiation, parameter specification, design command and design termination.

Description
This facility may be used to design isolated footings for user specified support joints. Once the support is specified, the program automatically identifies the support reaction(s) associated with the joint. All active load cases are checked and design is performed for the support reaction(s) that requires the maximum footing size. Parameters are available to control the design. Dowel bars and development lengths are also calculated and included in the design output.

Design Considerations
The STAAD-III isolated footing design is based on the following considerations.
1) The design reaction load may include concentrated load and biaxial moments.
2) The vertical reaction load is increased by 10% to account for the selfweight of the footing.
3) Footing slab size is rectangular. The ratio between the length and the width of the slab may be controlled by the user through a parameter.
4) Optional pedestal design is available.
5) Footings cannot be designed at supports where the reaction causes an uplift on the footings.

**Design Procedure**
The following sequential design procedure is followed:
1) Footing size is calculated on the basis of the load directly available from the analysis results (support reactions) and user specified Allowable Soil Pressure. No factor is used on the support reactions.
2) The footing size obtained from 1) and the FACTORED LOAD is utilized to calculate soil reactions.

FACTORED LOAD = ACTUAL REACTION X Parameter FFAC

Note that the user may provide a desired value for parameter FFAC.

3) Footing depth and reinforcement details are based on soil reactions calculated per 2) above.
4) Dowel bar requirements and development lengths are calculated and reported in the output.

Following parameters are available for footing design.

**Design Initiation**
**Purpose**
This command must be used to initiate the footing design.

**General Format:**

START FOOTING DESIGN

**Description**
This command initiates the footing design specifications. Without this command, no further footing design command will be recognized.

**Notes**
No footing design specification will be processed without this command.

**Footing Design Parameter Specification**
**Purpose**
This command is used to specify parameters that may be used to control the footing design.

**General Format:**
Description
Parameter-name refers to the parameters described in table (3).
f1 is the value of the parameter. Note that this value should be in the current units. The UNIT command is also accepted during any phase of footing design.

Notes
1) All parameter values must be provided in the current unit system.
2) For default values of parameters, refer to the parameter table (3).

Table (3) Footing Design Parameters

<table>
<thead>
<tr>
<th>Parameter name</th>
<th>Default Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>FY</strong></td>
<td>60,000 psi = 414 MPa</td>
<td>Yield strength for reinforcement steel. ($f_y$)</td>
</tr>
<tr>
<td><strong>FC</strong></td>
<td>3,000 psi = 20.7 MPa</td>
<td>Compressive Strength of Concrete. ($f'_c$)</td>
</tr>
<tr>
<td>CLEAR</td>
<td>3.0 in. = 76mm</td>
<td>Clear cover for slab reinforcement.</td>
</tr>
<tr>
<td>REINF</td>
<td>Number 9 bar</td>
<td>Main reinforcement bar size for slab design.</td>
</tr>
<tr>
<td>FFAC</td>
<td>1.0</td>
<td>Load factor for design.</td>
</tr>
<tr>
<td><strong>BC</strong></td>
<td>3000 psf = 143.6 MN/m$^2$</td>
<td>Soil bearing capacity.</td>
</tr>
<tr>
<td><strong>RATIO</strong></td>
<td>1.0</td>
<td>Ratio between slab sides.</td>
</tr>
</tbody>
</table>
| **TRACK**      | 1.0 | 1.0 = only numerical output is provided 
2.0 = numerical output and sketch provided |
| **DEPTH**      | Calculated by the program | The min. depth of the footing base slab. Program changes this value if required for design. |
| **S1, S2**     | Calculated by the program | Size of the footing base slab - S1 and S2 correspond to column sides YD and ZD respectively. Either S1 or S2 or both can be specified. If one is provided, the other will be calculated based on RATIO. If both are provided, RATIO will be ignored |
| **EMBEDMENT**  | 0.0 | The depth of the footing base from the support point of the column. |
### Footing Design Command

**Purpose**
This command must be used to execute the footing design.

**General Format:**

```
DESIGN FOOTING joint-list
```

**Description**
This command may be used to specify the joints for which the footing designs are required.

**Notes**
The output of this command may be controlled by the TRACK parameter (see table (3)). If TRACK is set to the default value of 1.0, only numerical output will be provided. If TRACK is set to 2.0, graphical output will be provided in addition.

**EXAMPLE**
```
START FOOTING DESIGN
CODE AMERICAN
UNIT KIP INCH
FY 45.0 JOINT 2
FY 60.0 JOINT 5
FC 3 ALL
RATIO 0.8 ALL
TRACK 2.0 ALL
PEDESTAL 1.0 ALL
UNIT KIP FEET
CLEAR 0.25
BC 5.20 JOINT 2
BC 5.00 JOINT 5
DESIGN FOOTING 1 2 3 5
END FOOTING DESIGN
```

### Footing Design Terminator

**Purpose**
This command must be used to terminate the footing design.

**General Format:**

```
END FOOTING DESIGN
```

**Description**
This command terminates the footing design.
Notes
If the footing design is not terminated, no further STAAD-III command will be recognized.

22- End Run Specification

Purpose
This command must be used to terminate the STAAD-III run.

General format:
FINISH

Description
This command should be provided as the last input command. This terminates a STAAD-III run.