ENGI 7704 Design of Steel Structures

Computer Analysis of Steel Structures by S-Frame
S-Frame Capabilities

- Automatic generation of trusses, multi-storey frames, arches, circles and spirals
- User friendly graphical interface
- International steel section and material database plus facilities to input tapered sections and custom sections using generic shape wizards or a simple graphical editor
- No practical limit on the size of the model
- Integrated Structural Steel Design (S-STEEL™) and Reinforced Concrete Design (S-CONCRETE™)
- Rigid and flexible diaphragm modeling
- Simple creation of panels (and holes) with automatic meshing and loading options
- Beam, truss, linear spring and inactive elements
- Buckling analysis
- Moving load analysis
- Staged construction
- Tension/compression only elements
- Many more advanced features
S-Frame Tutorial No.1

ANALYSIS & DESIGN OF STEEL TRUSS
The figure above shows a steel truss from several trusses supposed to cover a certain area. As shown, the truss has a cantilever part its span equals 4.0m. The proposed truss depth is 3.0m. The loads as shown, are concentrated at the truss joints. The values of its load case are shown. The horizontal bars are L55x55x3 and the vertical and diagonal bars are L45x45x3. Analyze this truss and investigate the if these sections meet the design criteria.
Engineering Problems

- Modeling
  - Geometry
  - Loading
- Analyzing
- Design
How to start S-Frame
Create New Model

Create a new 2D model
Setting up the model
Generation of Typical Structures
S-Frame Interface
Defining the Joints

Right Click

Specified number of joints can be automatically generated at equal distance

Specify the joint location
Defining the Joints

Total of 4 joints are generated.

Joint 1

3 joints are added to joint 1 with horizontal spacing of 4.0 m.
Defining the Joints

To generate the upper joints same procedure should be followed. First a joint at (0.0, 3.0) is added and then 3 joints are copied with horizontal distance of 4.0 m.

Total of 4 joints are generated

Joint 5

3 joints are added to joint 5 with horizontal spacing of 4.0 m
Zoom Structure Extents

To show all joints on the screen
The Generated Nodes
The coordinate system icon can be hidden.
Adding New Members

Members are added using the joints that have been already defined in the previous step.

Click here to enter member definition tool.
Members are defined between two joints.

Change this option to “Yes” so that the end joint of a member becomes the start joint of the next member. This option is useful when the members are defined next to each other.
Adding New Members

1: Click on first joint
2: Connect it to next joint
3: Because the “continue” option is selected another member is automatically drawn and it should be connected to next joint
4: When done, press ESC to cancel defining new members

Complete the Model
Assigning member type

Right click to activate the tool

Two node members can be any of these element types

These elements can be shown by different colors
Assigning member type

Legend shows that blue members are "Beam" elements. They should be changed to "Truss" elements.

Select the "Truss" member type
Assigning member type

1. Member type is set to “Truss”

2. Using the mouse, all the structure is selected by holding mouse left button and drawing a box around the structure. The members would then become red.
Moment Release

1. Select the moments to release

2. Select the members to apply the releases
Member Release
Defining Section Properties for the members

Section Color
Enter section properties

Click to access standard sections in the database
Importing Standard Sections

Section categories

Section properties

Section name
Importing Standard Sections

1. Click to select "Angles" to find desired sections from the drop-down menu.
2. Select the required section from the list.
3. Once selected, the properties of the standard section will be displayed in the Steel Section Properties dialog box.
Importing Standard Sections

- View of cross section and properties of the selected section
- Press to add the selected section to your list
- List of added sections
- Repeat the same steps to add the next section to the list
- When all the section are added press “Ok”
Importing Standard Sections

Assign “Blue” to first section and “Red” to the second one.
Assigning Section to the Members

The vertical and diagonal members become red to show they are assigned L45x45x3 section.

Legend shows all the sections are automatically assigned the first section in the list, i.e. L55x55x3.

Select second section L45x45x3

Using “Shift” key draw an intersecting line to select the vertical and diagonal members that should be assigned L45x45x3.
Assigning Material to the Members

All members by default are assigned “m1” material which is “Steel (US & Canada)”, therefore it is not needed to assign material to the members for this example.

If it is needed to assign another material to members of the model, first select the material name from the material list, then select the members to which the material should be assigned.
Add New Material

1. Right Click to open the tool

2. Choose a name for the new material

3. Enter material properties

4. Press to add the material
Assigning support to the joints

1. Right Click to open the tool

Choose the hinge to assign to joint 1 and press “Ok”

Possible supports in 2D, any of them can be chosen and assigned to a joint.
Assigning support to the joints

Hinge: movement is fixed at x and y direction

Activate the corresponding support (movement in y direction) to represent the roller and then click on joint 3 to which a roller should be assigned.

Click on joint 1 to be assigned the hinge.
The geometry of the model is now complete. In next step the model loading should be defined.

Remember to save your work from time to time
Define New Load Case

Click to add new Load Case
Define New Load Case

Choose a Load Case name, first case is “Self-Weight”

For Self-Weight load gravitational factor at y direction is set to -1.0

Press “Ok” to add the Load Case
Define New Load Case

Next Case is “Dead Load”

Gravitational factor should be set back to 0

Repeat this step to add other load cases:
Live Load: LL
Wind Load: WL

Press “Ok” to add the “DL” case to the load cases
Defining New Load Combination

Click to add new Load Combination
Defining New Load Combination

Load Combinations consist of a number of Load Cases multiplied by a Load Factor. In this example we use the following load combination:

\[ LC1 = 1.25 \text{DL} + 1.5 \text{LL} + 0.8 \text{WL} \]
Assign Nodal Load

1. Select Load Case from the case list menu
2. Select the load direction
3. Enter the load magnitude
4. Click on the nodes to which the load is applied
Nodal Loads on the Truss
When the model is ready, that is the geometry and loading is complete, it is possible to run the analysis to obtain the model deformations, member forces and stresses.

Analyze the Model
Analyze the Model

Analyze for both Load Cases and Load Combinations

Select the analysis type, choose “Linear Static” for this example

Press “Ok” to run the analysis
Analysis Report

S-FRAME FINITE ELEMENT ANALYSIS SOLVER (Ver 9.02)
COPYRIGHT (C) 1992-2009 SOFTEK SERVICES LTD.

2010-06-07, 15:01:46.151
Phase 1: memory management ...
Phase 2: program initialization ...
Phase 3: mesh optimization ...
Phase 4: model definition ...
Phase 5: element formulation ...
    dynamic memory allocated: 3 KB
Phase 6: assembly ...
Phase 7: solution and post-processing ...
    linear static analysis
    load case: 1
    load case: 2
    load case: 3
    load case: 4
    load combination: 1
Phase 8: cleaning up ...
    total number of FYI = 0
    total number of WARNINGS = 0
    total number of ERRORS = 0

2010-06-07, 15:01:47.776

Press any key to proceed to postprocessor
Graphical Results
Graphical Results

The analysis results can be viewed under any load case.

The loads of “Wind Load” case is shown.
Graphical Results

The results of “Load Combinations” can also be viewed.
Graphical Results

The deformed structure under the load combination
Graphical Results
Graphical Results

Check this box to show the numerical results on the diagrams
Graphical Results

Choose which data to be shown on the diagram
Graphical Results
Spreadsheets provide full access to the model geometry and loading. Any data can be edited in spreadsheets.
Spreadsheets
Spreadsheets

Press to switch between geometry spreadsheet and loading spreadsheets
Numerical Results

<table>
<thead>
<tr>
<th>Row</th>
<th>Joint</th>
<th>Load Comb</th>
<th>X - Tran</th>
<th>Y - Tran</th>
<th>Z - Rot</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1</td>
<td>1</td>
<td>0.0000</td>
<td>0.0000</td>
<td>0.0000</td>
</tr>
<tr>
<td>2</td>
<td>2</td>
<td>1</td>
<td>1.7690</td>
<td>-15.4863</td>
<td>0.0000</td>
</tr>
<tr>
<td>3</td>
<td>3</td>
<td>1</td>
<td>-1.3653</td>
<td>0.0000</td>
<td>0.0000</td>
</tr>
<tr>
<td>4</td>
<td>4</td>
<td>1</td>
<td>-1.4897</td>
<td>38.1986</td>
<td>0.0000</td>
</tr>
<tr>
<td>5</td>
<td>5</td>
<td>1</td>
<td>11.1225</td>
<td>-2.1548</td>
<td>0.0000</td>
</tr>
<tr>
<td>6</td>
<td>6</td>
<td>1</td>
<td>10.8732</td>
<td>-48.8544</td>
<td>0.0000</td>
</tr>
<tr>
<td>7</td>
<td>7</td>
<td>1</td>
<td>9.2637</td>
<td>-48.8519</td>
<td>0.0000</td>
</tr>
<tr>
<td>8</td>
<td>8</td>
<td>1</td>
<td>9.4133</td>
<td>-40.3505</td>
<td>0.0000</td>
</tr>
</tbody>
</table>
Numerical Results

Results are shown at different locations of a member.
### Numerical Results

![Image of numerical results interface]

#### Table of Results

<table>
<thead>
<tr>
<th>Row No.</th>
<th>Member No.</th>
<th>Load Comb No.</th>
<th>Station m</th>
<th>Axial (kN)</th>
<th>Shear (kN)</th>
<th>Moment (kN-m)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1</td>
<td>1</td>
<td>0.0000</td>
<td>-37.5057</td>
<td>0.0000</td>
<td>0.0000</td>
</tr>
<tr>
<td>2</td>
<td>1</td>
<td>1</td>
<td>3.0000</td>
<td>37.5057</td>
<td>0.0000</td>
<td>0.0000</td>
</tr>
<tr>
<td>3</td>
<td>2</td>
<td>1</td>
<td>0.0000</td>
<td>-4.0000</td>
<td>0.0000</td>
<td>0.0000</td>
</tr>
<tr>
<td>4</td>
<td>1</td>
<td>1</td>
<td>4.0000</td>
<td>-4.0000</td>
<td>0.0000</td>
<td>0.0000</td>
</tr>
<tr>
<td>5</td>
<td>3</td>
<td>1</td>
<td>0.0000</td>
<td>-25.8327</td>
<td>0.0000</td>
<td>0.0000</td>
</tr>
<tr>
<td>6</td>
<td>1</td>
<td>1</td>
<td>4.0000</td>
<td>-25.8327</td>
<td>0.0000</td>
<td>0.0000</td>
</tr>
<tr>
<td>7</td>
<td>4</td>
<td>1</td>
<td>0.0000</td>
<td>2.4000</td>
<td>0.0000</td>
<td>0.0000</td>
</tr>
<tr>
<td>8</td>
<td>1</td>
<td>1</td>
<td>4.0000</td>
<td>2.4000</td>
<td>0.0000</td>
<td>0.0000</td>
</tr>
<tr>
<td>9</td>
<td>5</td>
<td>1</td>
<td>0.0000</td>
<td>-37.4484</td>
<td>0.0000</td>
<td>0.0000</td>
</tr>
<tr>
<td>10</td>
<td>1</td>
<td>1</td>
<td>3.0000</td>
<td>-37.4484</td>
<td>0.0000</td>
<td>0.0000</td>
</tr>
<tr>
<td>11</td>
<td>6</td>
<td>1</td>
<td>0.0000</td>
<td>-50.1465</td>
<td>0.0000</td>
<td>0.0000</td>
</tr>
<tr>
<td>12</td>
<td>1</td>
<td>1</td>
<td>4.0000</td>
<td>-50.1465</td>
<td>0.0000</td>
<td>0.0000</td>
</tr>
<tr>
<td>13</td>
<td>7</td>
<td>1</td>
<td>0.0000</td>
<td>28.2327</td>
<td>0.0000</td>
<td>0.0000</td>
</tr>
<tr>
<td>14</td>
<td>1</td>
<td>1</td>
<td>4.0000</td>
<td>28.2327</td>
<td>0.0000</td>
<td>0.0000</td>
</tr>
<tr>
<td>15</td>
<td>8</td>
<td>1</td>
<td>0.0000</td>
<td>-27.3211</td>
<td>0.0000</td>
<td>0.0000</td>
</tr>
<tr>
<td>16</td>
<td>1</td>
<td>1</td>
<td>5.0000</td>
<td>-27.3211</td>
<td>0.0000</td>
<td>0.0000</td>
</tr>
<tr>
<td>17</td>
<td>9</td>
<td>1</td>
<td>0.0000</td>
<td>-60.5146</td>
<td>0.0000</td>
<td>0.0000</td>
</tr>
<tr>
<td>18</td>
<td>1</td>
<td>1</td>
<td>3.0000</td>
<td>-60.5146</td>
<td>0.0000</td>
<td>0.0000</td>
</tr>
<tr>
<td>19</td>
<td>10</td>
<td>1</td>
<td>0.0000</td>
<td>56.5716</td>
<td>0.0000</td>
<td>0.0000</td>
</tr>
<tr>
<td>20</td>
<td>1</td>
<td>1</td>
<td>4.0000</td>
<td>56.5716</td>
<td>0.0000</td>
<td>0.0000</td>
</tr>
</tbody>
</table>
### Numerical Results

**S-Frame Software Interface:**

- **Displacements:**
- **Velocities:**
- **Accelerations:**
- **Reactions:**
- **Member Forces:**
- **Wall Forces:**
- **Member Stresses:**
- **Shell Stresses:**
- **Stoery:**
- **Frequencies:**
- **Mode Shapes:**
- **Total Mass:**
- **Active Mass:**
- **Buckling Factors:**
- **Buckling Modes:**
- **Envelopes:**

#### Member Forces Details:

<table>
<thead>
<tr>
<th>Load Comb No</th>
<th>Station m</th>
<th>Axial kN</th>
<th>Shear kN</th>
<th>Moment kN-m</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0.0000</td>
<td>-37.5657</td>
<td>0.0000</td>
<td>0.0000</td>
</tr>
<tr>
<td>1</td>
<td>3.0000</td>
<td>-37.5657</td>
<td>0.0000</td>
<td>0.0000</td>
</tr>
<tr>
<td>1</td>
<td>6.0000</td>
<td>0.0000</td>
<td>-4.8000</td>
<td>0.0000</td>
</tr>
<tr>
<td>1</td>
<td>9.0000</td>
<td>0.0000</td>
<td>-4.8000</td>
<td>0.0000</td>
</tr>
<tr>
<td>1</td>
<td>12.0000</td>
<td>0.0000</td>
<td>-25.8327</td>
<td>0.0000</td>
</tr>
<tr>
<td>1</td>
<td>15.0000</td>
<td>0.0000</td>
<td>-25.8327</td>
<td>0.0000</td>
</tr>
<tr>
<td>1</td>
<td>18.0000</td>
<td>0.0000</td>
<td>2.4000</td>
<td>0.0000</td>
</tr>
<tr>
<td>1</td>
<td>21.0000</td>
<td>0.0000</td>
<td>2.4000</td>
<td>0.0000</td>
</tr>
</tbody>
</table>

**Select Load Type:**

- **Primary Load Cases**
- **Groups**
- **Tools**

**S-Frame Software Menu:**

- **File**
- **Edit**
- **View**
- **Format**
- **Settings**
- **Options**
- **Run**
- **Window**
- **Help**

**Additional Features:**

- **Cad Details Design**
- **Reactions**
- **Member Forces**
- **Normal Stresses**
- **Shear Stresses**
- **Wall Forces**
- **Base Shear**
- **Wall Forces**
Design
Design Module: S-Steel
Always remember to select all the members
Code Check

The “Code Check” window examines if the current sections of the model meet the design code provision.
Utilizations Ratios
Design Report
Member: 12

S-FRAME Section is 45X45X3

Member is part of group: Section 2

Note: Neglecting axial<1.0 kN, shear<1.0 kN, moment<1.0 kNm

Note: Member in brace frame Angle Gamma is -90.0 degrees

Load Case 1 Self-Weight (Unloaded)

Section classification (εf = 250 MPa);

| Section Class | 4 |

Governing geometrical slenderness ratio

L / μ ≤ \[ \frac{336}{300} \]

1.120

Load Case 2 DL (Compression)

Governing geometrical slenderness ratio

L / μ ≤ \[ \frac{336}{200} \]

1.679

Axial Load (N):

450

Factored Compressive Resistance Check:

\[ \frac{C_t}{C_n} = \frac{\phi A f (1 + k_2)^{5+\gamma}}{\phi A (17 MPa)} \]

\[ = \frac{45.0}{4.06} > 10 \]

Load Case 3 LL (Compression)

Governing geometrical slenderness ratio

L / μ ≤ \[ \frac{336}{200} \]

1.679

Axial Load (N):

575

Factored Compressive Resistance Check:

\[ \frac{C_t}{C_n} = \frac{\phi A f (1 + k_2)^{5+\gamma}}{\phi A (17 MPa)} \]

\[ = \frac{67.5}{4.06} > 10 \]

Load Case 4 WL (Compression)

Governing geometrical slenderness ratio

L / μ ≤ \[ \frac{336}{200} \]

1.679

Axial Load (N):

500

Factored Compressive Resistance Check:

\[ \frac{C_t}{C_n} = \frac{\phi A f (1 + k_2)^{5+\gamma}}{\phi A (17 MPa)} \]

\[ = \frac{16.5}{4.06} > 10 \]

Design Code: CAN/CSA S16-01
Steel Table: Canadian (CISC)
### Spreadsheet Window

![Spreadsheet Window](image)

<table>
<thead>
<tr>
<th>Row No.</th>
<th>Member No.</th>
<th>Design Status</th>
<th>Code Check Status</th>
<th>Member Length (m)</th>
<th>Group Name</th>
<th>S-FRAME Section</th>
<th>User Override Section</th>
<th>Design Proposed Section</th>
<th>Design Scratch Pad Override</th>
<th>Code Check Scratch Pad Results</th>
<th>Design Section Shape</th>
<th>Design Spec Criteria</th>
<th>Yield Strength (MPa)</th>
<th>Ultimate Strength (N)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1</td>
<td>Valid</td>
<td>Valid</td>
<td>3.00</td>
<td>Section 2</td>
<td>L45 x 45 x 3</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>Angle</td>
<td>Weight</td>
<td>350.0</td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>2</td>
<td>Valid</td>
<td>Valid</td>
<td>4.00</td>
<td>Section 1</td>
<td>L55 x 55 x 3</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>Angle</td>
<td>Weight</td>
<td>350.0</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>3</td>
<td>Valid</td>
<td>Valid</td>
<td>4.00</td>
<td>Section 1</td>
<td>L55 x 55 x 3</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>Angle</td>
<td>Weight</td>
<td>350.0</td>
<td></td>
</tr>
<tr>
<td>4</td>
<td>4</td>
<td>Valid</td>
<td>Valid</td>
<td>4.00</td>
<td>Section 1</td>
<td>L55 x 55 x 3</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>Angle</td>
<td>Weight</td>
<td>350.0</td>
<td></td>
</tr>
<tr>
<td>5</td>
<td>5</td>
<td>Valid</td>
<td>Valid</td>
<td>3.00</td>
<td>Section 2</td>
<td>L45 x 45 x 3</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>Angle</td>
<td>Weight</td>
<td>350.0</td>
<td></td>
</tr>
<tr>
<td>6</td>
<td>6</td>
<td>Valid</td>
<td>Valid</td>
<td>4.00</td>
<td>Section 1</td>
<td>L55 x 55 x 3</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>Angle</td>
<td>Weight</td>
<td>350.0</td>
<td></td>
</tr>
<tr>
<td>7</td>
<td>7</td>
<td>Valid</td>
<td>Valid</td>
<td>4.00</td>
<td>Section 1</td>
<td>L55 x 55 x 3</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>Angle</td>
<td>Weight</td>
<td>350.0</td>
<td></td>
</tr>
<tr>
<td>8</td>
<td>8</td>
<td>Valid</td>
<td>Valid</td>
<td>4.00</td>
<td>Section 1</td>
<td>L55 x 55 x 3</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>Angle</td>
<td>Weight</td>
<td>350.0</td>
<td></td>
</tr>
<tr>
<td>9</td>
<td>9</td>
<td>Valid</td>
<td>Valid</td>
<td>5.00</td>
<td>Section 2</td>
<td>L45 x 45 x 3</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>Angle</td>
<td>Weight</td>
<td>350.0</td>
<td></td>
</tr>
<tr>
<td>10</td>
<td>10</td>
<td>Valid</td>
<td>Valid</td>
<td>3.00</td>
<td>Section 2</td>
<td>L45 x 45 x 3</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>Angle</td>
<td>Weight</td>
<td>350.0</td>
<td></td>
</tr>
<tr>
<td>11</td>
<td>11</td>
<td>Valid</td>
<td>Valid</td>
<td>5.00</td>
<td>Section 2</td>
<td>L45 x 45 x 3</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>Angle</td>
<td>Weight</td>
<td>350.0</td>
<td></td>
</tr>
<tr>
<td>12</td>
<td>12</td>
<td>Valid</td>
<td>Valid</td>
<td>3.00</td>
<td>Section 2</td>
<td>L45 x 45 x 3</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>Angle</td>
<td>Weight</td>
<td>350.0</td>
<td></td>
</tr>
<tr>
<td>13</td>
<td>13</td>
<td>Valid</td>
<td>Valid</td>
<td>5.00</td>
<td>Section 2</td>
<td>L45 x 45 x 3</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>Angle</td>
<td>Weight</td>
<td>350.0</td>
<td></td>
</tr>
</tbody>
</table>
Design

Section that can be used in the design

- File
- Edit
- View
- Select
- Run
- Options
- Window
- Help

Section Shapes:
- Equal Legs
- Unequal Legs

Section Type:
- L200X200X30
- L200X200X25
- L200X200X20
- L200X200X16
- L200X200X13
- L200X200X10
- L150X150X20
- L150X150X16
- L150X150X13
- L150X150X10

Exclude:
- Sections not matching Tolsa
- Sections not matching Revit

Selection Criteria:
- Weight
- Cost
- Surface Area
- Depth
- Width
- Ix
- Iy
- Ix

Group Name:

Help

CAN/CSA S16-01
Design Loads

Select all load cases and load combinations to considered in the design.
Reanalyze the model after the “Design” as new sections are assigned to the members.
All the sections pass the “Code Check” after the design.