DESIGN PROJECT FOR STEEL AND TIMBER DESIGN COURSE (CIV 312)
& SELF-CODED DESIGN SOFTWARE FOR SECTION SELECTION

Introduction:

This report is about a design project that was done at the end of the Steel and Timber Design course. The goal of the project was to develop the core structural elements of a 5-storey steel building. I have learned how to apply the concepts written in a code (CISC "Handbook of Steel Construction" & CSA S16-14) to a real building. The design project was done in a group of 5. I was responsible for choosing members and constituting the model. I used an analysis software called S-Frame since it was in the requirements of the project. I also verified my results using SAP2000. In addition to those, I wrote a design software to choose the appropriate members. The software included, and the model included are entirely my own work.

We were only allowed to use W sections in this project. The model was designed to be simply supported except the moment resisting frame (MRF). The dimensions of the building layout were given to us as well as where to place the concentrically braced frame (CBF). Our version of the S-Frame did not have the feature to check against the code and suggest a member, and my version of SAP2000 did not have Canadian codes. So, I coded a software to choose the members. I also did several hand calculations to verify the answers.

The software I coded looks into a spreadsheet saved in a text file that contains the section properties of almost every W section that is available in Canada, goes over the calculations required by the code for each section and then stores the results in memory. It is programmed to choose the least amount steel for the columns, for the beams and for the beam-columns. It takes beam or beam-column height into consideration as well. As soon as the program finds a better section, it ignores the previous section properties. On the upcoming parts of this entry, there are more details about the software. The software is also available at: https://www.koraleren.com/programming/c/#C1.

Below, the 3D appearance of the building's structural elements with an additional planar view with the loads is available.

This project was my first chance to practice my profession. The foundation design, mould plans, electrical and mechanical cladding were out of the scope of this project. This project improved my design, analysis and teamwork skills and prepared me for the real work experience.

Building that had to be designed:

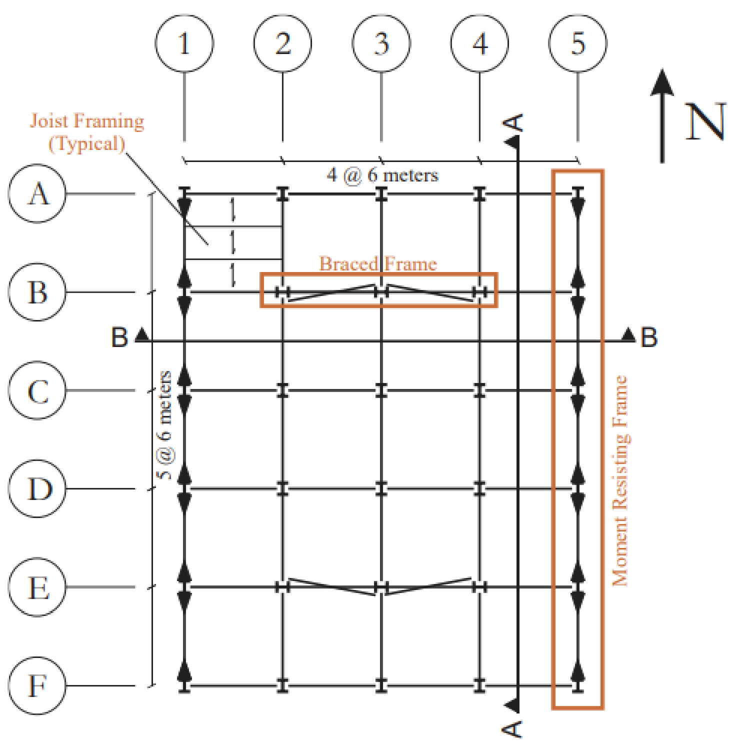
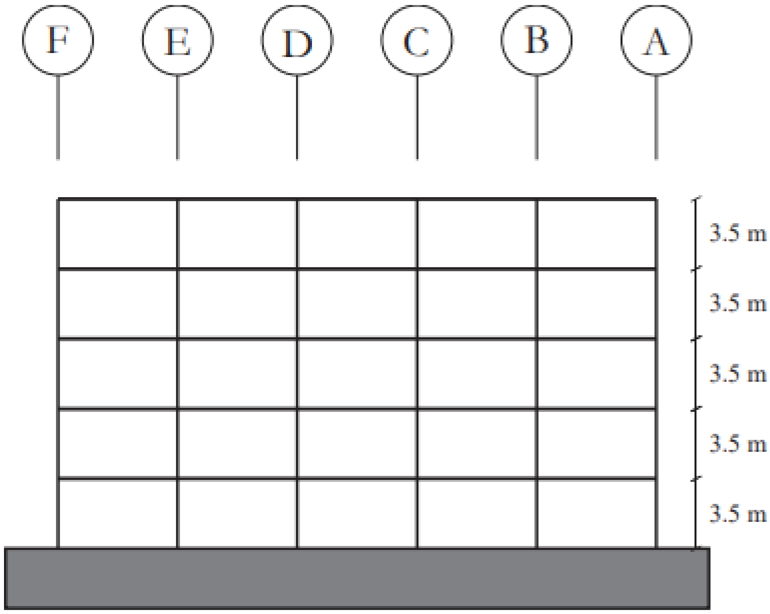
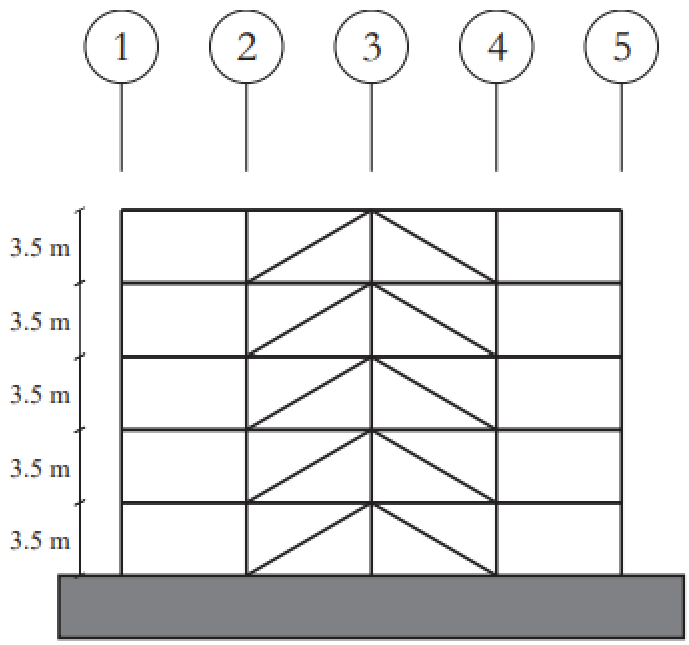


Figure A.1: Building Plan



Section A - A

Figure A.2: Moment Resisting Frame

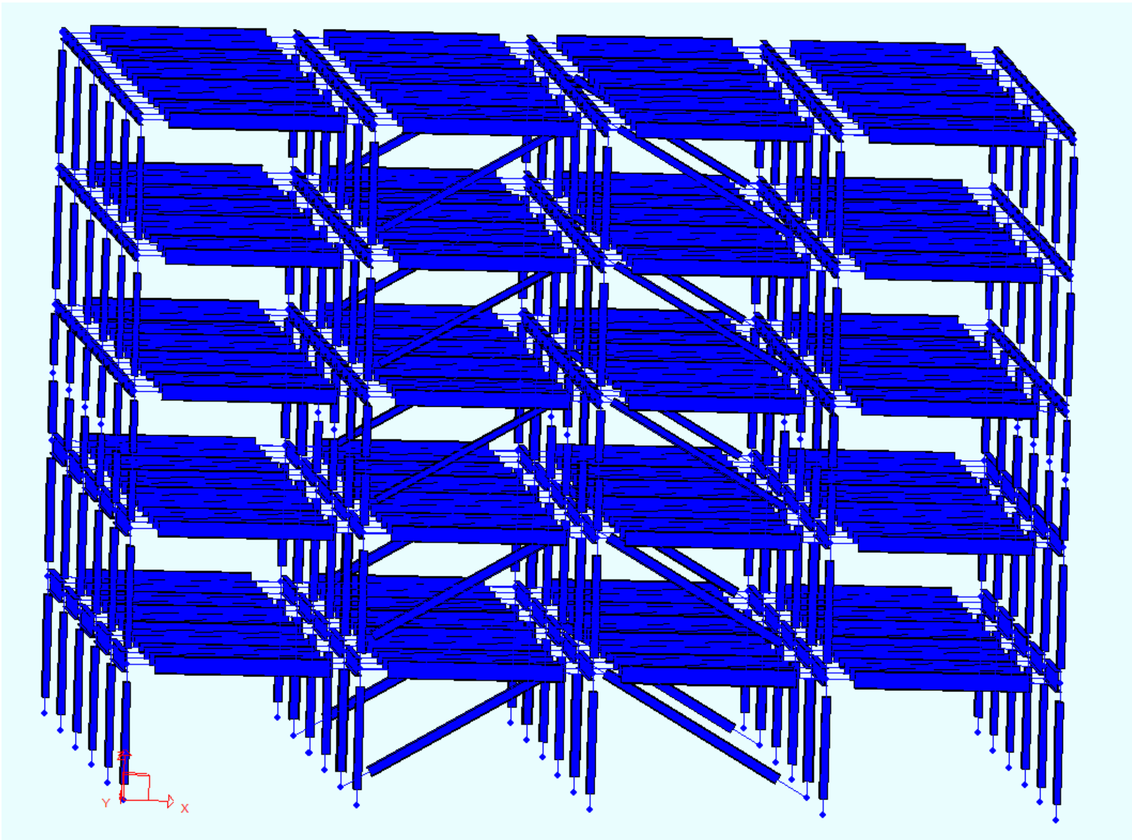


Section B -B

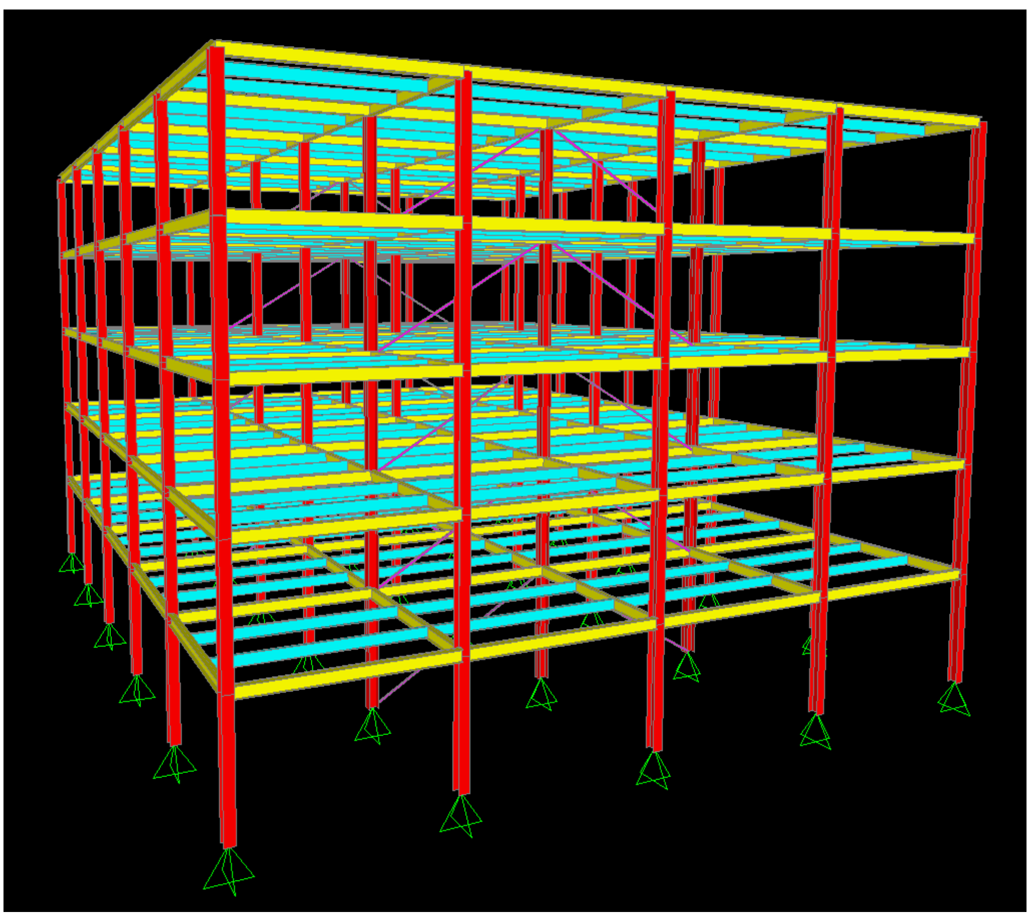
Figure A.3: Concentrically Braced Frame (CBF)

3D View of the computer model of the building:

a) S-Frame:



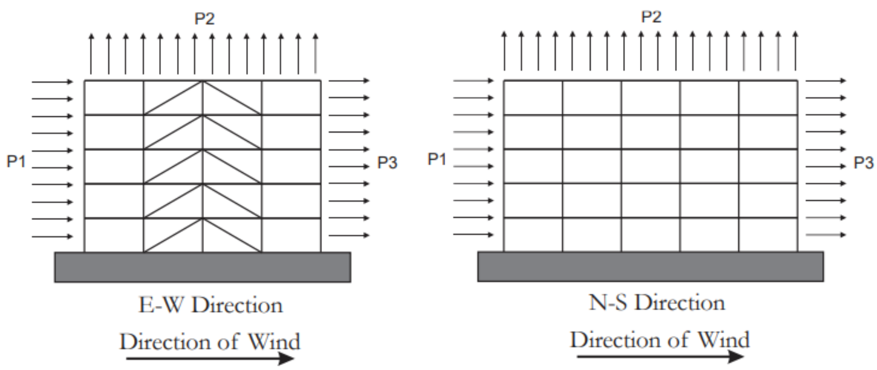
b) Sap2000:



Calculated loading and load combinations:

Based on the given information from the assignment and NBCC 2015, the following loading applied to the structure:

Dead Load Roof: 3.59 kPa
Dead Load 1.2.3.4. Floor: 5.12 kPa
Live Load 1. Floor: 4.8 kPa
Live Load 2.3.4. Floor: 2.4 kPa
Live Load Roof: 1 kPa
Snow Load Roof: 1.82 kPa
Wind Loading: P1 = 1.5 kPa, P2 = 0.5 kPa, P3 = 1.2 kPa



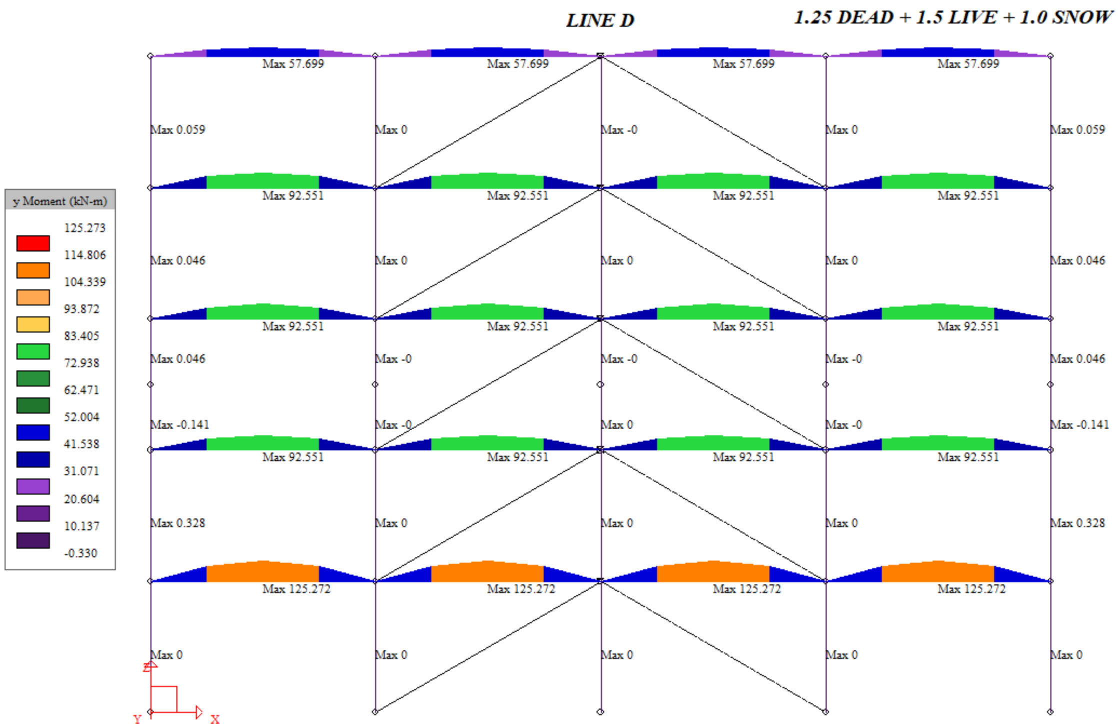
Load combinations used based on NBCC 2015:

DL = Dead Load | LLF = Live load in 1.2.3.4. floors | LLR = Live load in roof | SL = Snow Load | WLNS = Wind Load in North-South Direction (includes uplift) | WLEW = Wind Load in East-West Direction (includes uplift)

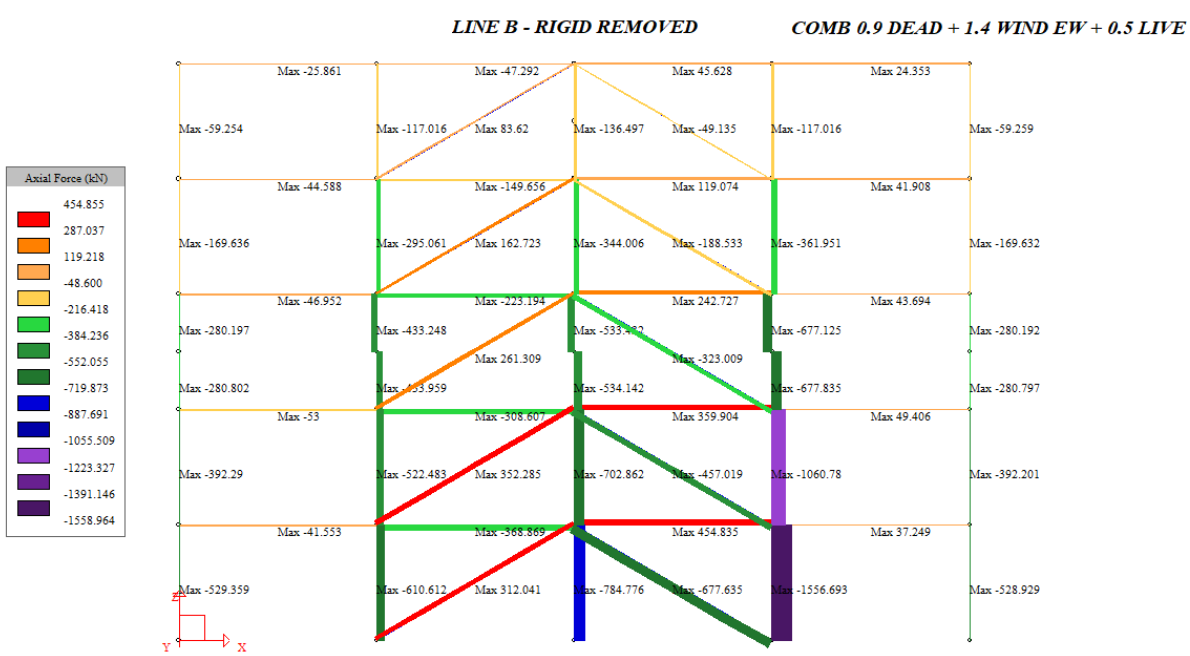
1.4 DL
1.25 DL + 1.5 LLF + 1.5 LLR
1.25 DL + 1.5 SL
0.9 DL + 1.4 WLNS
0.9 DL + 1.4 WLEW
1.25 DL + 1.5 LLF + 1.0 SL
0.9 DL + 1.5 LLF + 1.5 LLR + 0.4 WLNS
0.9 DL + 1.5 LLF + 1.5 LLR + 0.4 WLEW
1.25 DL + 1.5 SL + 1.0 LLF
0.9 DL + 1.5 SL + 0.4 WLNS
0.9 DL + 1.5 SL + 0.4 WLEW
0.9 DL + 1.4 WLNS + 0.5 LLF + 0.5 LLR
0.9 DL + 1.4 WLEW + 0.5 LLF + 0.5 LLR
0.9 DL + 1.4 WLNS + 0.5 SL
0.9 DL + 1.4 WLEW + 0.5 SL

<u>Analysis results based on preliminary conservative members:</u>

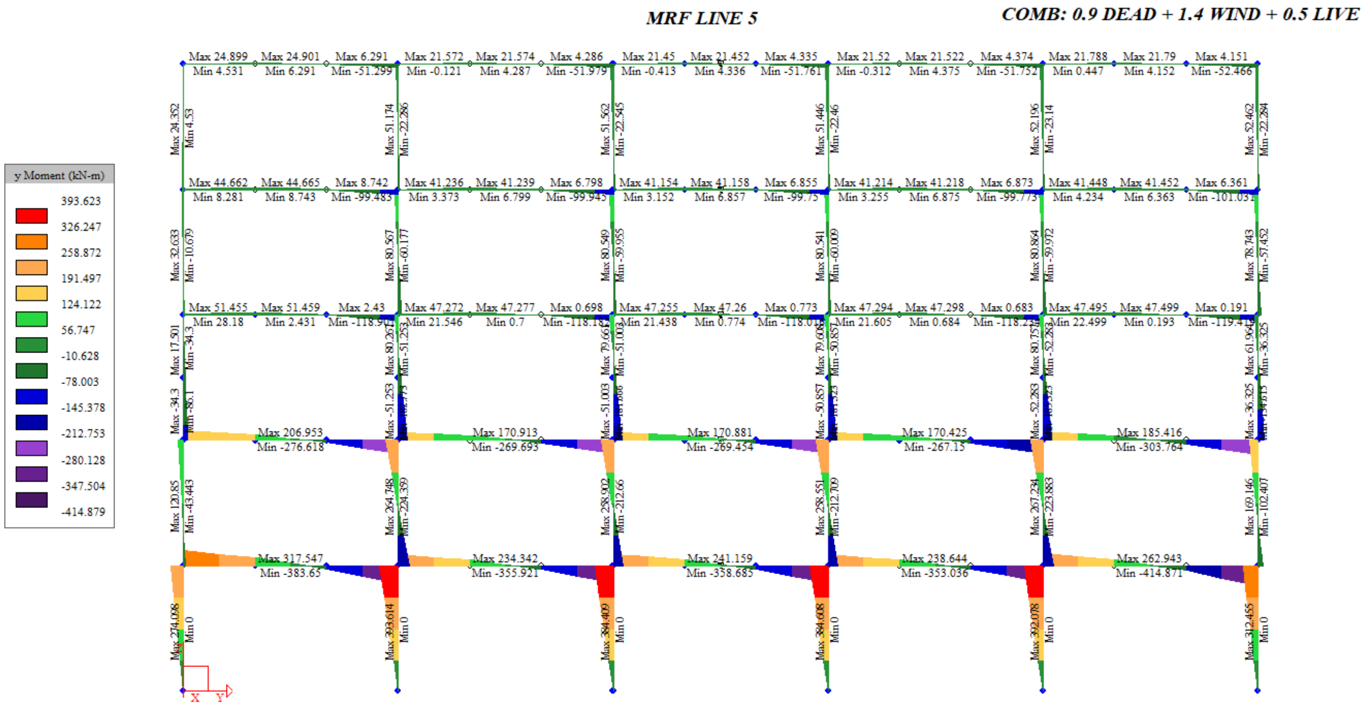
-Moments at Line D for the dominating combination:



-Axial Forces at Line B when rigid diaphragm constraint removed:



-Moments acting on the Moment Resisting Frame at Line 5:



<u>Choosing the most appropriate members by using the least amount of steel possible based on the analysis results and the code using my design software:</u>

My design software doesn't have a graphical user interface, looks like a software from 80s, but it is fast and gets the job done. At that time, I was only comfortable in c language, so it was written in c.

It consists of 8 files and their function can be understood from their names:

BeamColumnDesign.c -> Designs the beam column
BeamColumnDesign.h
BeamDesign.c -> Designs the beam
BeamDesign.h
ColumnDesign.c -> Designs the column
ColumnDesign.h
main.c (Main file is for the menu and navigating trough functions)
Sections.txt (Contains almost all the W steel cross-sections available in Canada)

-Main file source code:

```
#include < stdio.h > #include < conio.h > #include < stdlib.h > #include < math.h >

  #include "BeamColumnDesign.h"#
include "BeamDesign.h"#
include "ColumnDesign.h"

int main(void) {
  int choice = 0, i;
  while (choice != '4') {
    printf("\n       DESIGN SOFTWARE KORAL\n");
    for (i = 0; i < 33; i++)
      printf("*");
    printf("\nPRESS 1 FOR COLUMN DESIGN");
    printf("\nPRESS 2 FOR BEAM DESIGN");
    printf("\nPRESS 3 FOR BEAM-COLUMN DESIGN");
    printf("\nPRESS 4 TO EXIT");
    printf("\nEnter Your Choice: ");
    choice = getche();
    switch (choice) {
    case '1':
```

```
      printf("\nYOU SELECTED COLUMN DESIGN\n\n\n");
      ColumnDesign();
      printf("\n\n\n");
      system("pause");
      system("cls");
      break;
    case '2':
      printf("\nYOU SELECTED BEAM DESIGN\n\n\n");
      BeamDesign();
      printf("\n\n\n");
      system("pause");
      system("cls");
      break;
    case '3':
      printf("\nYOU SELECTED BEAM COLUMN DESIGN\n\n\n");
      BeamColumnDesign();
      printf("\n\n\n");
      system("pause");
      system("cls");
      break;
    case '4':
      printf("\nYOU SELECTED EXIT\n\n\n");
      break;
    default:
      printf("\n\nINVALID SELECTION...Please try again\n");
    }
  }
  return EXIT_SUCCESS;
```

Some parts of beam-column design source code:

```
void BeamColumnDesign (void)
{
     /**********************FILE READER************************/
     FILE *fp;
  double sections1D [2842];
  double value;
  int i = -1;

  if ((fp = fopen ("Sections.txt", "r")) == NULL)
  {
      printf("Can't find the Sections file called Sections");
      return;
  }


  while (!feof (fp) && fscanf (fp, "%lf", &value) && i++ < 2841)
    sections1D [i] = value;

  fclose (fp);

  int j;
  double sections [203][14];

  for (j=0;j<203;j++)
    for (i=0;i<14;i++)
        sections[j][i]=sections1D[i+(14*(j))];
/**********************FILE READER***************************/

    int flangeclass;
    int webclass;
    int shapeclass;
    double Mf, Cf, L, Lu, slendernessratiox, slendernessratioy;
    double Mr, Mmax, Ma, Mb, Mc, w2, Mu, G = 77000, PI = 3.141593;
    int braced;
    double Cr0, Mr0;
    double U1, w1, Ce;
    double crosssectionalcheck = 0, overallmemberstrength = 0, LTBstrength = 0, ADDcheck = 0;
    double Crx, Cry, Fe, Lambda, n = 1.34;
    double massselect = 10000;
    int sectionselect = 0, sectionselectholder = 0;
```

….. (Some code in between)

Example of Cross-sectional Strength Check based on steel code:

```
if (braced == 1)
    {
        printf("CROSS SECTIONAL STRENGTH CHECK:\n");
```

```
        Cr0 = 0.90*sections[j][2]*350*0.001;
        printf("Cr0 = %lf kN\n", Cr0);

        printf("Mr0 (knM) calculation:\n");

    if (shapeclass == 1 || shapeclass == 2)
    {
        Mr0 = 0.90*sections[j][5]*350*0.000001*1000;
        printf ("Mr0 = %lf kNm\n", Mr0);
    }
    else if (shapeclass == 3)
    {
        Mr0 = 0.90*sections[j][4]*350*0.000001*1000;
        printf ("Mr0 = %lf kNm\n", Mr0);
    }

        printf("U1 calculation:\n");

        Ce = pow(PI,2)*(200000)*(sections[j][3])*(1000000)*(0.001)/pow(L,2);

        U1 = w1/(1-Cf/Ce);

        if (U1 < 1)
            U1 = 1;

        printf ("U1 = %lf\n", U1);

        if ((shapeclass == 1 || shapeclass == 2) && (Cf/Cr0 + (0.85*U1*Mf)/Mr0) <= 1)
        {
            crosssectionalcheck = (Cf/Cr0 + (0.85*U1*Mf)/Mr0);
            printf("%lf <= 1 - Cross Sectional Check OK\n\n", crosssectionalcheck);
        }
        else if (shapeclass == 3 && (Cf/Cr0 + (U1*Mf)/Mr0) <= 1)
        {
            crosssectionalcheck = (Cf/Cr0 + (U1*Mf)/Mr0);
            printf("%lf <= 1 - Cross Sectional Check OK\n\n", crosssectionalcheck);
        }
        else
        {
            printf("Cross Sectional Check FAIL\n\n");
            crosssectionalcheck = 2;
        }

    }
```

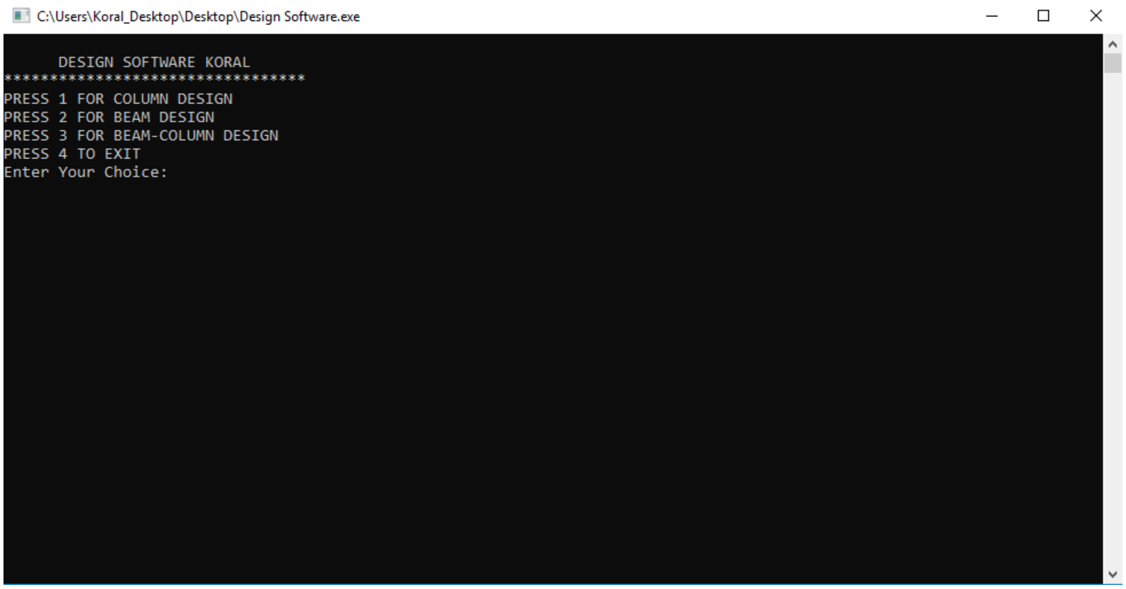
..... (Some code in between)

Choosing the section:

```
if (crosssectionalcheck<=1 /*&& sections[j][0]>400*/ && overallmemberstrength<=1 && LTBstrength<=1 && ADDcheck<=1 &&
shapeclass<4 && massselect>=sections[j][1])
    {
        sectionselect = j;
        massselect = sections[j][1];
    }

    printf("\n\nSection selected = W%lfx%lf\n\n", sections[sectionselect][0], sections[sectionselect][1]);
```

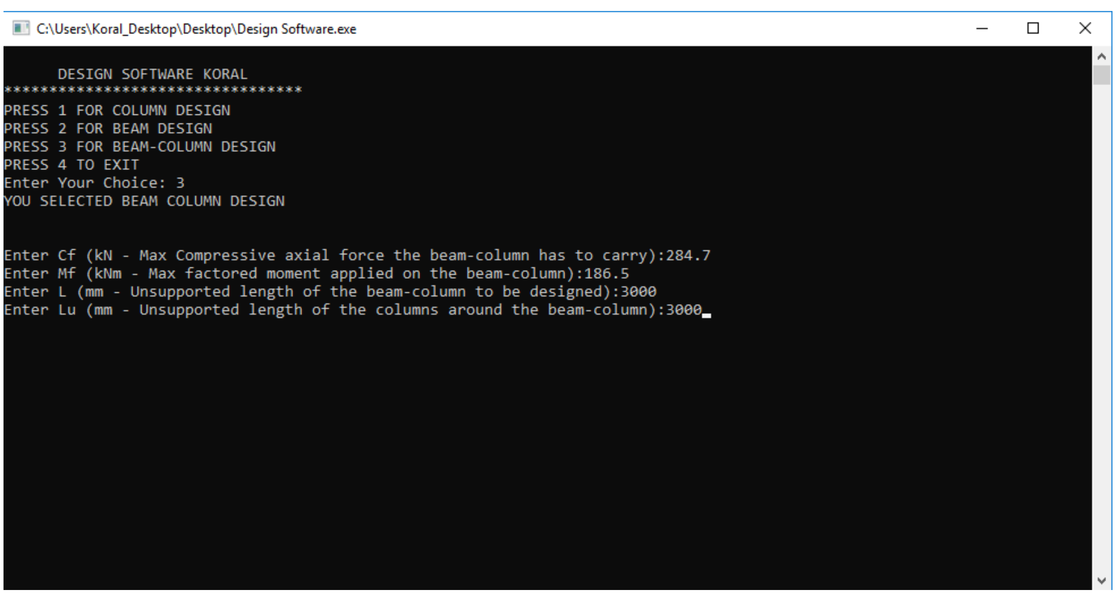
Let's say I need to design an I-beam column that has to resist a maximum factored compressive axial force (Cf) of 284.7 kN and a maximum factored moment (Mf) of 186.5 kNm based on previous analysis.

When the software is run, this screen comes:



I want to design a beam-column, so I press 3. Then it asks me to enter Cf, so I type 284.7 and hit enter. Then it asks me to enter Mf, so I type 186.5 and hit enter. Then it asks me the unsupported length of the beam-column. I go back to my model on S-Frame or SAP2000 and check the length, type and hit enter. Let's say 3000 mm. I type 3000 mm and hit enter. Then it asks the unsupported length of the columns around the beam-column element that I'm trying to design. I again look at my model and enter those. For entry, if they are all similar just one value needs to be typed before hitting enter. If they are not similar, then going from left to right and then from bottom to top should be entered with a space in between. 3000 2000 2000 3000 then enter for example. Let's assume all are the same so I type 3000 and hit enter.

The screen now looks like this:



After, I must give some preliminary boundary condition information. Let's say my beam-column is fixed both ends and not connected to ground. So, I type 4 and hit enter. This is for preliminary calculations to reduce iterations. Later, the stiffness of the columns will be accounted.

C:\Users\Koral_Desktop\Desktop\Design Software.exe

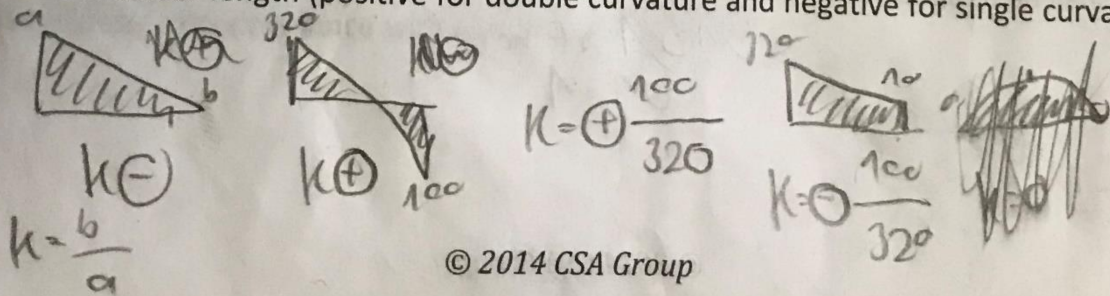
```
     DESIGN SOFTWARE KORAL
*********************************
PRESS 1 FOR COLUMN DESIGN
PRESS 2 FOR BEAM DESIGN
PRESS 3 FOR BEAM-COLUMN DESIGN
PRESS 4 TO EXIT
Enter Your Choice: 3
YOU SELECTED BEAM COLUMN DESIGN


Enter Cf (kN - Max Compressive axial force the beam-column has to carry):284.7
Enter Mf (kNm - Max factored moment applied on the beam-column):186.5
Enter L (mm - Unsupported length of the beam-column to be designed):3000
Enter Lu (mm - Unsupported length of the columns around the beam-column):3000

The following calculations are for sidesway prevented frames:
If BOTH ends are PINNED and not connected to ground PRESS 1:
If LEFT or LOWER end is pinned to the GROUND and RIGHT or UPPER END IS PINNED PRESS 2:
If LEFT or LOWER end is pinned to the GROUND and RIGHT or UPPER END IS FIXED PRESS 3:
If BOTH ends are FIXED and not connected to ground PRESS 4:
4
```

After It asks me for w1. (I look at the code, see below)

13.8.5 Values of ω_1

Unless otherwise determined by analysis, the following values shall be used for ω_1:

a) for members not subjected to transverse loads between supports:
$\omega_1 = 0.6 - 0.4\kappa \geq 0.4$ No load or moment between supports
where
κ = ratio of the smaller factored moment to the larger factored moment at opposite ends of the member length (positive for double curvature and negative for single curvature)



1-66 © 2014 CSA Group June 2014

S16-14 Design of steel structures

b) for members subjected to distributed loads or a series of point loads between supports:
$\omega_1 = 1.0$

c) for members subjected to a concentrated load or moment between supports:
$\omega_1 = 0.85$
For the purpose of design, members subjected to a concentrated load or moment between supports (e.g., segmented columns) may be considered to be divided into segments at the points of load (or moment) application. Each segment shall then be treated as a member that depends on its own flexural stiffness to prevent sidesway in the plane of bending considered and ω_1 shall be taken as 0.85. In calculating the slenderness ratio for use in Clause 13.8, the total length of the compression member shall be used.

Note: *For references to more exact methods often justified for crane-supporting columns and similar applications, see Annex C.*

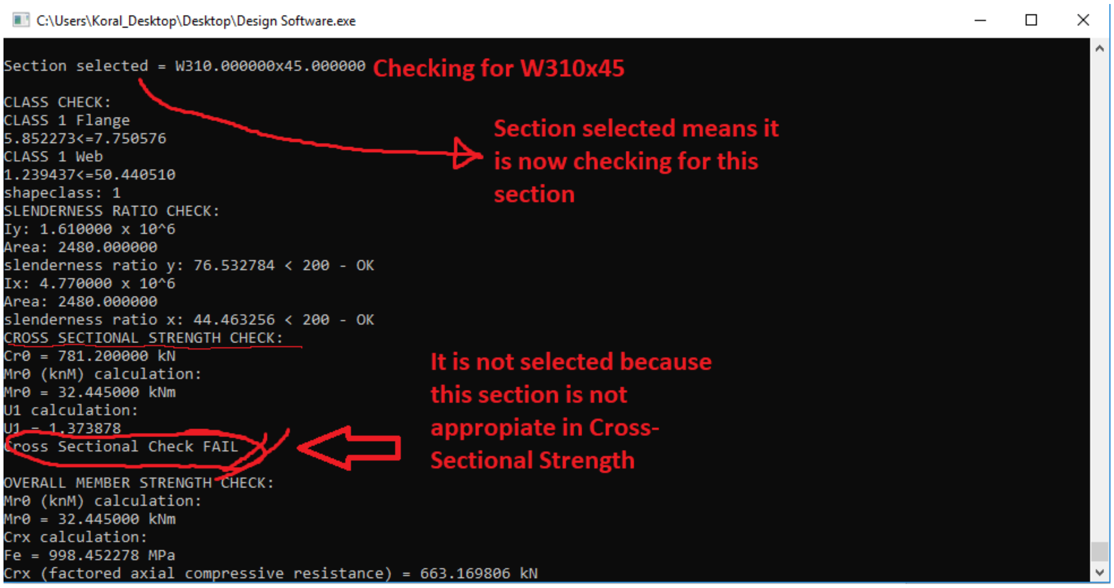
Let’s say, my member is subjected to distributed loads so w1 = 1. I type 1 and hit enter. Later, I need to calculate w2. For that, I type my maximum moment Mf, Ma, Mb, Mc. Since my moment is parabolic because of distributed loading, I enter 186.5, 139.88, 186.5, 139.88 for the Mf, Ma, Mb and Mc respectively.

```
C:\Users\Koral_Desktop\Desktop\Design Software.exe
PRESS 1 FOR COLUMN DESIGN
PRESS 2 FOR BEAM DESIGN
PRESS 3 FOR BEAM-COLUMN DESIGN
PRESS 4 TO EXIT
Enter Your Choice: 3
YOU SELECTED BEAM COLUMN DESIGN


Enter Cf (kN - Max Compressive axial force the beam-column has to carry):284.7
Enter Mf (kNm - Max factored moment applied on the beam-column):186.5
Enter L (mm - Unsupported length of the beam-column to be designed):3000
Enter Lu (mm - Unsupported length of the columns around the beam-column):3000

The following calculations are for sidesway prevented frames:
If BOTH ends are PINNED and not connected to ground PRESS 1:
If LEFT or LOWER end is pinned to the GROUND and RIGHT or UPPER END IS PINNED PRESS 2:
If LEFT or LOWER end is pinned to the GROUND and RIGHT or UPPER END IS FIXED PRESS 3:
If BOTH ends are FIXED and not connected to ground PRESS 4:
4
K = 0.650000

Enter w1:1
w2 for LATERALLY UNSUPPORTED LENGTH Mr (knM) CALCULATION:
Enter maximum moment (Mmax) kNm - no negative-: 186.5
Enter moment at 1/4 of unbraced segment (Ma) kNm: 139.88
Enter moment at 2/4 of unbraced segment (Mb) kNm: 186.5
Enter moment at 3/4 of unbraced segment (Mc) kNm: 139.88
```

Then it asks me whether the frame is braced or not. Let’s say it is braced so I press 1. Now, My first set of data entry is finished. The screen looks like this:

```
C:\Users\Koral_Desktop\Desktop\Design Software.exe
Enter Your Choice: 3
YOU SELECTED BEAM COLUMN DESIGN


Enter Cf (kN - Max Compressive axial force the beam-column has to carry):284.7
Enter Mf (kNm - Max factored moment applied on the beam-column):186.5
Enter L (mm - Unsupported length of the beam-column to be designed):3000
Enter Lu (mm - Unsupported length of the columns around the beam-column):3000

The following calculations are for sidesway prevented frames:
If BOTH ends are PINNED and not connected to ground PRESS 1:
If LEFT or LOWER end is pinned to the GROUND and RIGHT or UPPER END IS PINNED PRESS 2:
If LEFT or LOWER end is pinned to the GROUND and RIGHT or UPPER END IS FIXED PRESS 3:
If BOTH ends are FIXED and not connected to ground PRESS 4:
4
K = 0.650000

Enter w1:1
w2 for LATERALLY UNSUPPORTED LENGTH Mr (knM) CALCULATION:
Enter maximum moment (Mmax) kNm - no negative-: 186.5
Enter moment at 1/4 of unbraced segment (Ma) kNm: 139.88
Enter moment at 2/4 of unbraced segment (Mb) kNm: 186.5
Enter moment at 3/4 of unbraced segment (Mc) kNm: 139.88

w2: 1.131356

Is the frame braced?
PRESS 1 FOR YES
PRESS 2 FOR NO: 1
Press any key to continue . . .
```

When I hit a key, it selects a preliminary section. I made it in such a way so that it prints all the iterations and calculations for the sections that are not selected. This way, it is possible to determine why they are not selected.




```
Section selected = W310.000000x45.000000

CLASS CHECK:
CLASS 1 Flange
5.852273<=7.750576
CLASS 1 Web
1.239437<=50.440510
shapeclass: 1
SLENDERNESS RATIO CHECK:
Iy: 1.610000 x 10^6
Area: 2480.000000
slenderness ratio y: 76.532784 < 200 - OK
Ix: 4.770000 x 10^6
Area: 2480.000000
slenderness ratio x: 44.463256 < 200 - OK
CROSS SECTIONAL STRENGTH CHECK:
Cr0 = 781.200000 kN
Mr0 (knM) calculation:
Mr0 = 32.445000 kNm
U1 calculation:
U1 = 1.373878
Cross Sectional Check FAIL

OVERALL MEMBER STRENGTH CHECK:
Mr0 (knM) calculation:
Mr0 = 32.445000 kNm
Crx calculation:
Fe = 998.452278 MPa
Crx (factored axial compressive resistance) = 663.169806 kN
```

At the very bottom, when the iterations complete (takes about a second because it prints all the data). It selects a preliminary section and now asks more information about the stiffnesses of the connections. In the Sections.txt file, it can find all the sectional properties for the sections when user enters the section depth and nominal weight. So, here all I need to enter is those for the sections asked. Let's say all columns and beams are W310x45 for simplicity. I enter that to every field it asks me. After, it asks me whether ends are not supported or not. Both ends are fixed and supported in this case so I type 2 two times and hit enter.

```
Enter column or beam sections:
Enter upper column or beam section W ? x ?:
?1:
310
?2:45

Section chosen: W310.000000x45.000000

Is one end not supported?
PRESS 1 FOR YES
PRESS 2 FOR NO: 2
Are two ends not supported?
PRESS 1 FOR YES
PRESS 2 FOR NO: 2
Enter lower column or beam section W ? x ?:
?1:310
?2:45

Section chosen: W310.000000x45.000000

K = 0.542270

Press any key to continue . . .
```

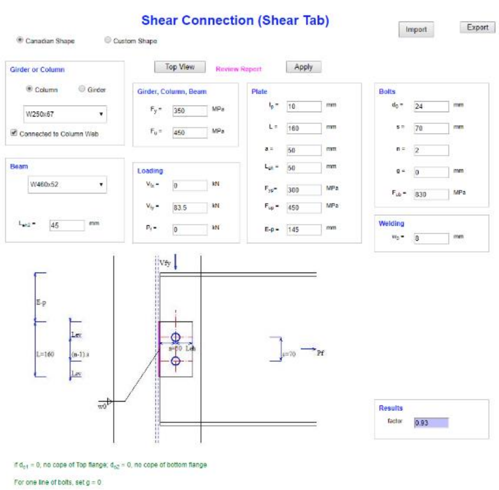
As you can see, it calculated a K value different than initial 0.65. Now it will reiterate to find a better section if it can. In this case it chose W310x45 as the final section. It prints all the calculations for it and why it passes. But it also again prints for the sections that fail and why they fail resisting loads. After reviewing results, I can go back to S-Frame or Sap2000 and safely assign that section. After a key is pressed, it goes back to main menu from which another member can be designed.

C:\Users\Koral_Desktop\Desktop\Design Software.exe

```
Mr0 = 223.020000 kNm
Crx calculation:
Fe = 13003.381931 MPa
Crx (factored axial compressive resistance) = 1781.889342 kN

U1 = 1.013259
0.880010 <= 1 - Overall Member Strength Check OK

LATERAL TORSIONAL BUCKLING CHECK:
Mr0 (knM) calculation:
Mr0 = 223.020000 kNm
Cry calculation:
Fe = 1120.755197 MPa
Cry (factored axial compressive resistance) = 1554.459919 kN

Mr (knM) calculation:

Mu: 371.421229 kNm

Mr: 208.562107 kNm
U1 = 1.013259
0.953314 <= 1 - Lateral Torsional Buckling Check OK

ADDITIONAL CHECK:
0.894218 <= 1 - Additional Check OK



Press any key to continue . . .
```

Connection design by using an online commercial software (checked by hand calculations):

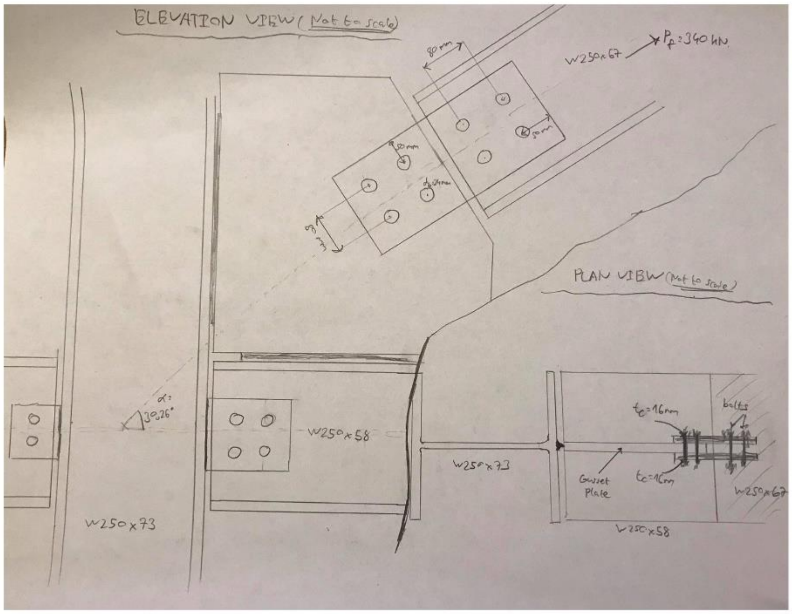
Example software design:



Sample of some checks done by hand:
Bolt Group Shear Unity Check:
xm = (cb - 1) . g / 2 = = … mm
ym = (n - 1). s / 2 = = … mm
Sum(x2+y2) = … mm^2
Rfx = (Vfy . eb). ym / Sum(x2 + y2) = … kN
Rfy = (Vfy . eb). xm / Sum(x2 + y2) + Vfy / (cb . n) = … kN
Horizontal axial force on one bolt
Rh = … kN
Vfb = … kN
Bolt shear Resistance
Factored shear resistance per bolt: =… kN,
Plate bearing Resistance
Vrb2 = … kN,
Beam web bearing Resistance
Vrb3 = … kN,
Bolt Tear Out of Plate Edge
Vrb4 = …kN,
Bolt Tear Out of beam flange Edge
Bolt vertical edge distance for beam (assumed): … mm
Bolt edge distance for beam: … mm
Vrb5 = … kN,
Factored shear resistance per bolt: Vrb = Min. [Vrb1, Vrb2, Vrb3, Vrb4, Vrb5] = Min. […,…,…,…,…] =… kN

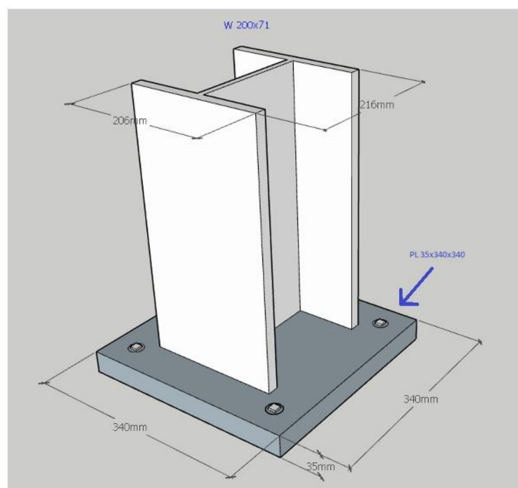
Final connection design and member selection:

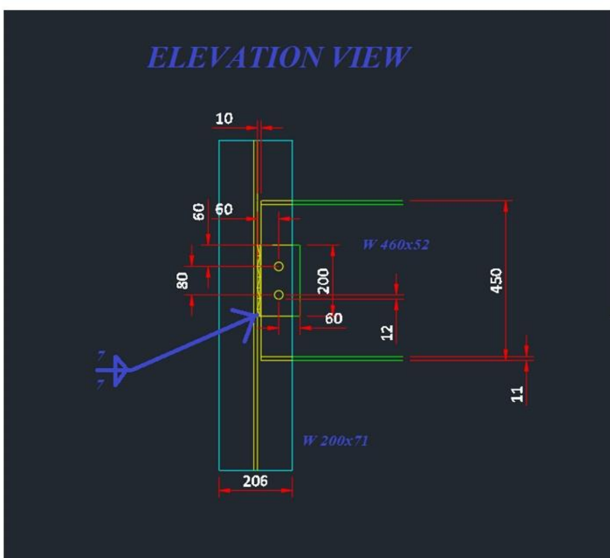
CONNECTION DETAILING

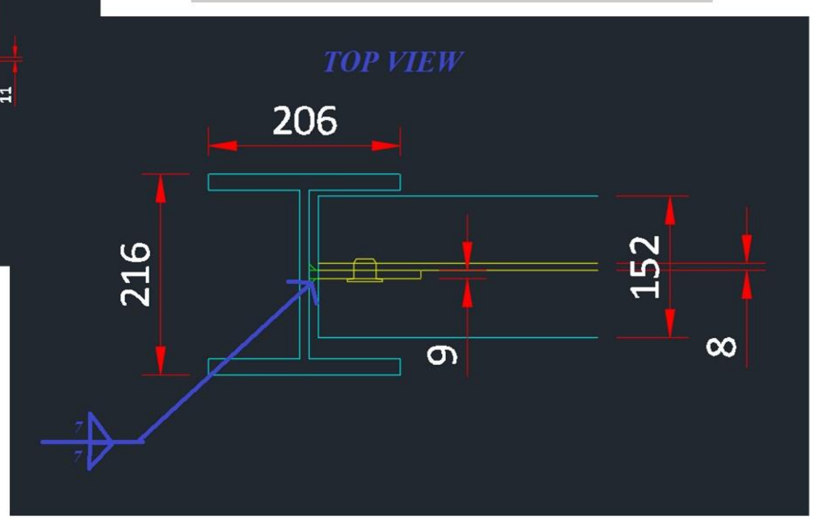


Bracing Connection

Base Plate Detailing

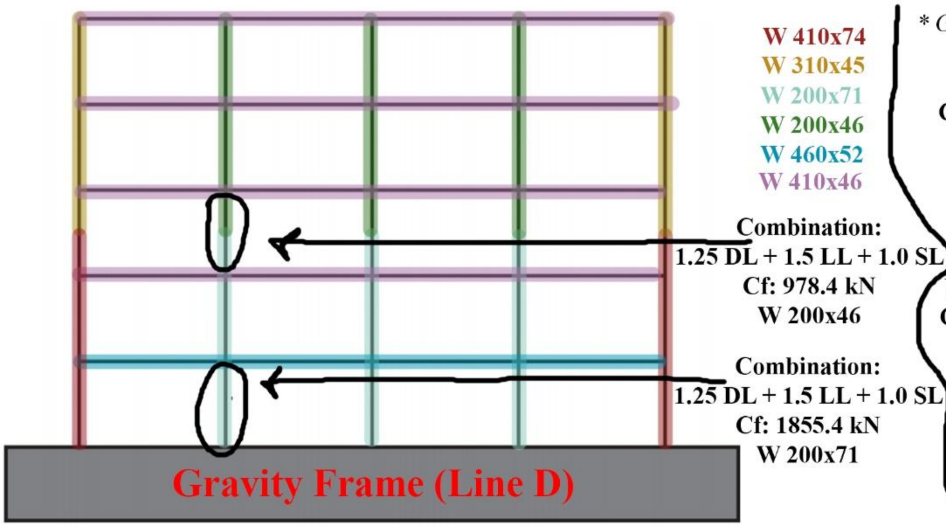






Shear Tab Connection Detailing

FINAL MEMBER SELECTION



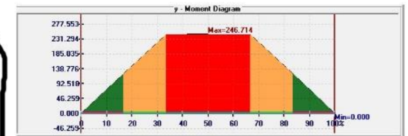
** Gravity beams selected based on the following moment from beams on line 2,3 and 4*

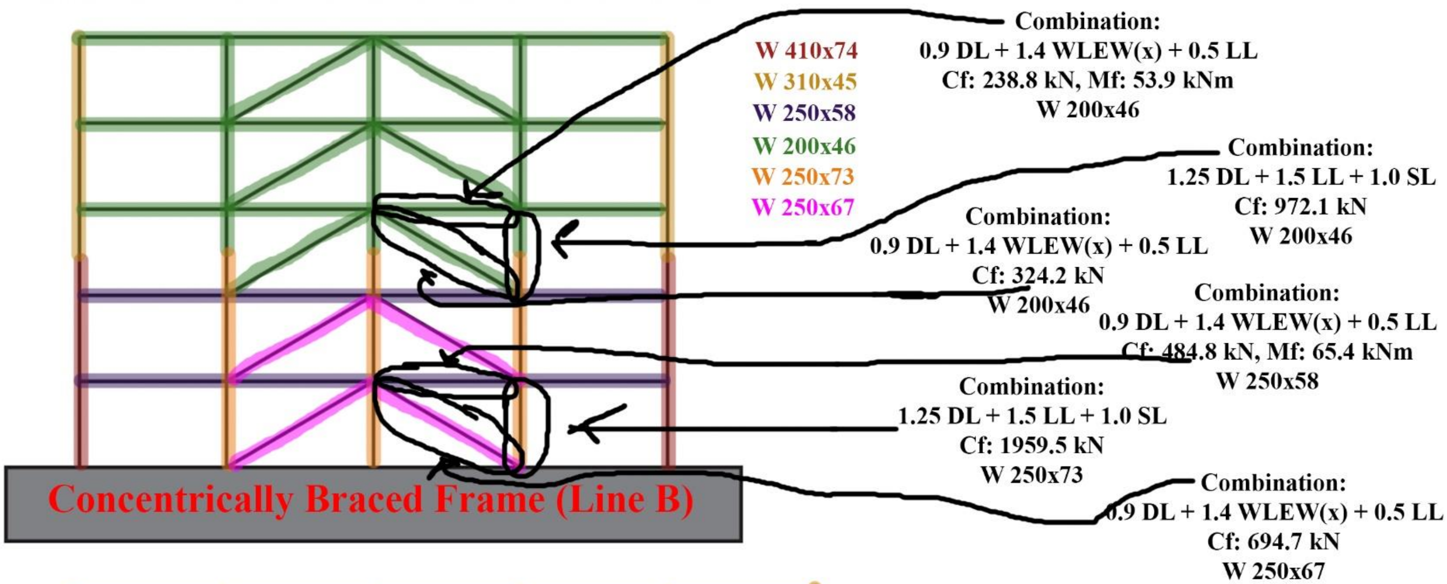
1st Floor Gravity Beams: W 460x52

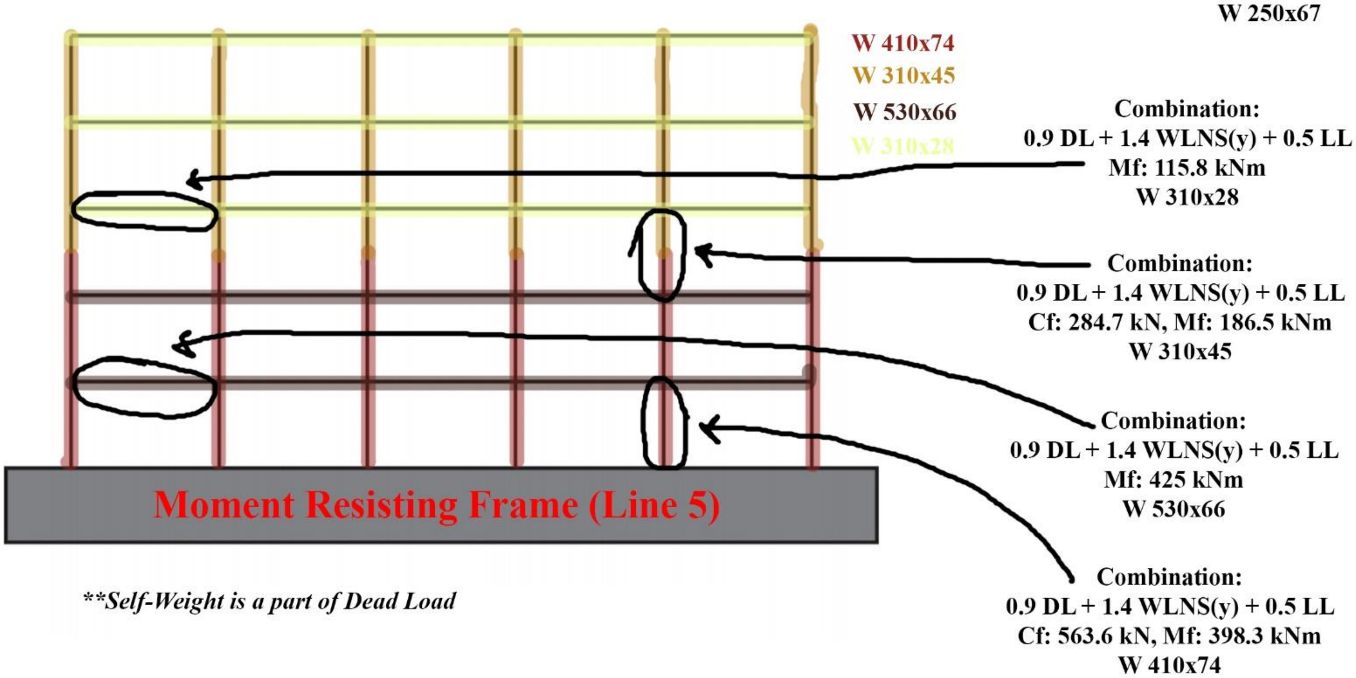
Combination: 1.25 DL + 1.5 LL + 1.0 SL

Upper Floor Gravity Beams: W 410x46

Combination: 1.25 DL + 1.5 LL + 1.0 SL







***Self-Weight is a part of Dead Load*