ASDIP STRUCTURAL SOFTWARE

ASDIP Steel

User's Manual

ASDIP Steel

STRUCTURAL ENGINEERING SOFTWARE

Table of Contents

Welcome to ASDIP Steel

Frequently the design process involves an iterative procedure of selecting preliminary proportions of structural elements, and then checking the suitability of this solution, otherwise new member properties are used until the algorithm converges in an optimum design. This repetitive and tedious procedure may become time and effort consuming.

ASDIP Steel is a collection of calculation modules that carefully combine the latest building code provisions and proved design and analysis methodologies to perform many of the cumbersome calculations most commonly used in any structural design office. ***ASDIP Steel*** is an integrated system that combines the flexibility of Windows Forms to effortlessly develop either an optimized design or a quick investigation.

All the modules have been assembled to help the designer obtain specific results from procedures common to structural steel design. However, they cannot replace the judgment of an experienced engineer who must select the structural types and appropriate loads, and interpret the results. ***ASDIP Steel*** fully complies with the AISC 360-05. The Base Plate Design is based on the AISC Design Guides # 1 Second Edition. The anchorage design fully complies with the ACI 318-08 Appendix D "Anchoring to Concrete".

License Agreement

IMPORTANT - READ BEFORE COPYING, INSTALLING OR USING.

Do not copy, install, or use this software until you have carefully read this agreement, which provides the following terms and conditions for its use. Copying, installing, or otherwise using the software indicates that you have read, understand and accept the terms of this Agreement. If you do not accept this Agreement, then return the software and accompanying items to us within 30 days of purchase for a refund.

GRANT OF LICENSE

The software and data accompanying this License (the "Software") and related documentation are licensed (not sold) to you by ASDIP Structural Software. You own the CD or any other media on which the Software is recorded, but ASDIP Structural Software Steels full and complete title to the Software and accompanying documentation, and all intellectual property rights therein.

Subject to the License Restrictions below, you are granted a personal, non-exclusive license to use one copy of the Software and any updates thereof on a single computer at a time. You may authenticate your license in a second computer, but not in a third, since only two authentications are granted. You may make one archival or backup copy of the software for your own use. However, because the Software is protected by the copyright laws, it is illegal: to make additional copies or otherwise duplicate the Software by any other means (including electronic transmission); to give copies to another person; or, to modify, adapt, translate, rent, sub-license, loan, resell for profit, distribute, create derivative works based upon or network the Software or any part thereof.

LICENSE RESTRICTIONS

The software contains copyrighted material, trade secrets and other proprietary material protected by United States copyright laws and international treaty provisions. You may not reverse-assemble, reverse-compile, or otherwise reverse-engineer any of the Software components. You will not remove any copyright notices from the Software

screens and printouts. ASDIP Structural Software does not grant any express or implied right to you under patents, copyrights, trademarks, or trade secret information.

DISCLAIMER OF WARRANTIES

The Software is provided "AS IS" without warranty of any kind. If the media on which the Software is furnished are found to be defective in material or workmanship under normal use within a period of thirty (30) days from the date of receipt, ASDIP Structural Software's entire liability and your exclusive remedy shall be the replacement of the media. This offer is void if the media defect results from accident, abuse, or misapplication.

To the maximum extent permitted by law, ASDIP Structural Software disclaims all other warranties, either express or implied, including, but not limited to, implied warranties or merchantability and fitness for a particular purpose, with regard to the Software and the accompanying materials. This limited warranty gives you specific legal rights. You may have others that vary from state/jurisdiction to state/jurisdiction.

LIMITATION OF LIABILITY

To the maximum extent permitted by applicable law, in no event shall ASDIP Structural Software be liable for any damages whatsoever (including, without limitation, damages for loss of business profits, business interruption, loss of business information, or any other loss) arising out of the use of or inability to use this Software. ASDIP Structural Software makes no warranties, either expressed or implied, as to the quality and performance of the Software, that the calculations contained in the Software will meet your requirements, or that the operation of the Software will be uninterrupted or error free. You agree that the Software is intended to aid you in performing engineering-oriented mathematical calculations, and that the results obtained from the Software do not necessarily constitute an acceptable engineering design or analysis. You agree that results of the Software shall be reviewed by persons possessing experience and thorough understanding of the engineering principles that apply or might apply to the usage of and results of the Software.

In no event shall ASDIP Structural Software's total liability to you for all damages, losses, and causes of action (whether in contract or otherwise) exceed the amount paid by you for the Software.

How to Start a New Project

When you start ***ASDIP Steel*** the Project Manager pops up. From there, you may either create a new project or open an existing one. If you are working in a project and just want to create a new one select File | New from the menu bar. ***ASDIP Steel*** will ask you to confirm, in case that you haven't saved your previous project. The new project will have the calculation tree empty. From there you may start creating calculations.

How to Open an Existing Project

ASDIP Steel projects are saved with the extension .rdp. To open an existing project, in the Project Manager select File | Open from the menu. A new dialog box will pop up to let you specify the location of the requested file. When an existing project is retrieved, all the information and calculations saved with that project will be retrieved as well, so that all the information regarding that specific project is brought up.

How to Save a Project

ASDIP Steel projects are saved with the extension .rdp. To save a project, in the Project Manager select File | Save from the menu. If the project has been saved previously, it will be saved directly, otherwise a new dialog box will pop up to let you specify the location of the file. When a project is saved, all the settings and calculations will be saved as well, so that all the information regarding that specific project will be ready to come up when you open the project in the future. When you save the project the file name, which is also the project name, will be appended to the title in the Project Manager.

How to Specify the Units for a Project

In ***ASDIP Steel*** you may work with any of the following three units systems:

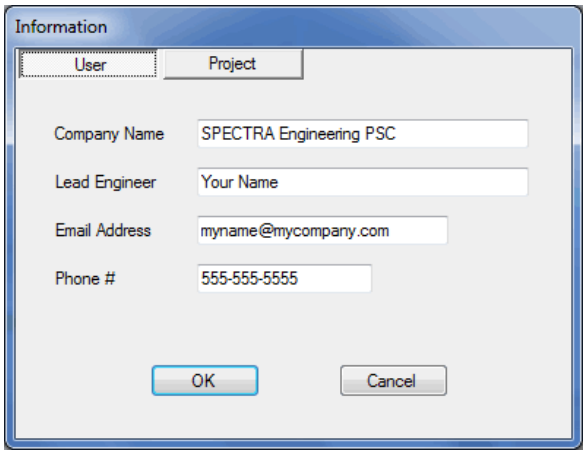
- US - Imperial units, customarily used in the United States (in, ft, kip, ksi)
- SI - The International Standard system of units (cm, m, N, MPa)
- ME - Metric units, mostly used in Latin America (cm, m, Tn, Kg/cm^2)

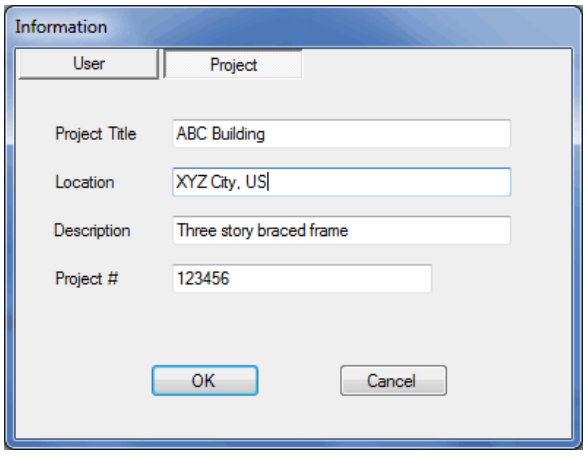
You may specify the desired units system in two different ways:

- Directly in the Settings Menu of the Project Manager. This will affect the units of the whole project. The default is US.
- In the Design Menu of the individual calculations. This setting will affect only that specific calculation, and it may be useful if for some reason you need to calculate something in other units different to the rest of the project.

How to Enter User and Project Information

When you work in a project, it's a good practice to provide all the information regarding the designer and the project itself. ***ASDIP Steel*** provides two dialog boxes for this purpose in the Settings Menu of the Project Manager, as shown. This information is also saved with the rest of the calculations that belong to the project.



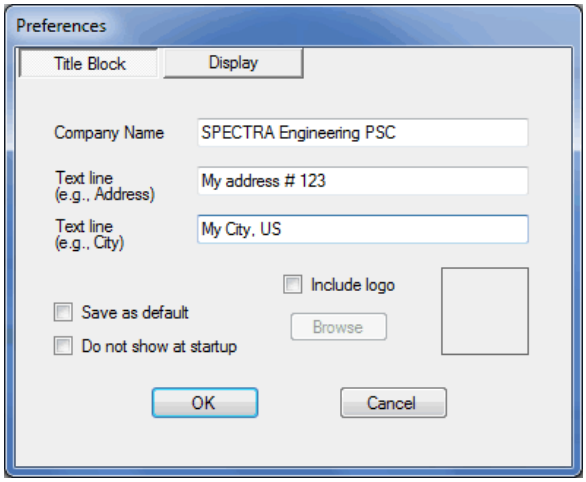


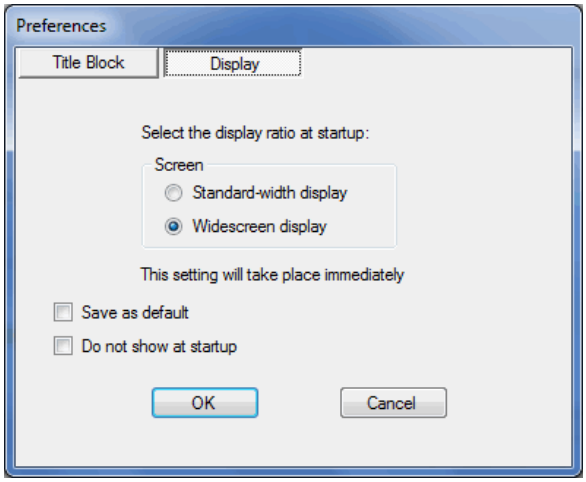
How to Set Preferences for a Project

ASDIP Steel allows you set project preferences in order to customize your program and improve your experience. Although not mandatory, these settings will affect the way the program looks and the way your Reports will be generated. You may set the preferences in the Settings Menu of the Project Manager, and they consist of two dialog boxes, as shown below. One is for the Title Block definition that will be included in the Report and the other is to specify the display screen All this information may be saved as default, so you don't need to enter it again in the future. When you start ***ASDIP Steel*** the Preferences dialog box shows up. You may instruct the program not to show it at startup by checking mark the corresponding check box.

The Title Block information is mostly used as a header in the Report. and it consists of basic information about your company or any other text you may want to include in your Report. You may also specify your company logo.

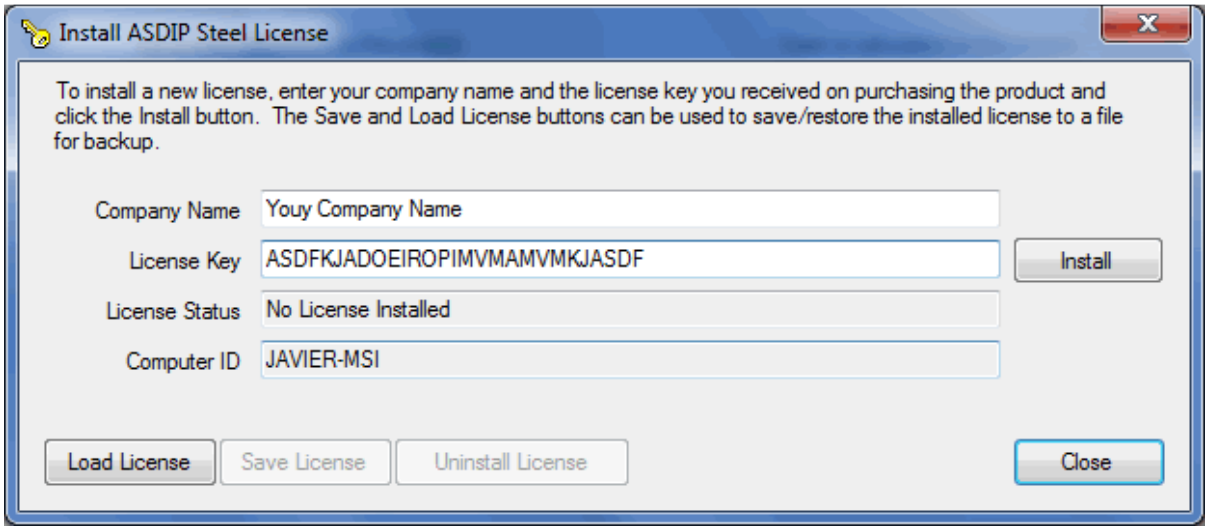
The display screen may be either Standard-width or Widescreen display, depending of your monitor. Basically, the Standard-width display fits the screens with a 4:3 ratio such as the common desktop monitors. Widescreen display works better in flat screens with 16:10 ratio such as many laptops and some LCD monitors.





How to Authenticate the License

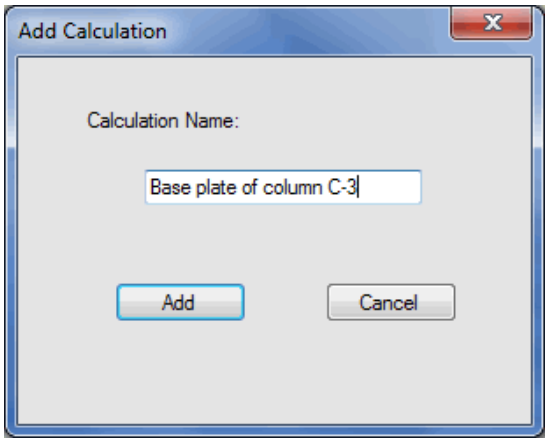
When you download and start using ***ASDIP Steel***, what you are actually running is an Evaluation version, which is fully functional for 15 days. The only limitation during this period is that you cannot printout your calculations and you cannot save your project, otherwise it is the full version that you will permanently have if you authenticate your license. To do so, you need to order the program by visiting our web site www.asdipsoft.com. Then an email will be sent to you with a license key, which will convert your Evaluation version into the Full version. To install your license either select Settings | License in the Project Manager or select *Install License* in the Evaluation dialog box at startup. The dialog box shown below will show up.

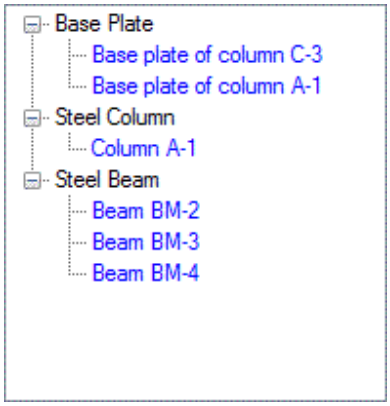


In this dialog box you enter your company name and the license key that you just received in your email inbox. When you click the *Install* button the program connects to our web server and authenticates the license for you. Note that you have to have an internet connection to do the authentication. Once the authenticated license is installed on your computer the application can be run without contacting the authentication service. Basically **your license will be linked to your computer name**, not to your hardware. This will allow you to upgrade your hardware without invalidating your license, provided you keep your computer name. Two authentications are granted with your license, which means that you may authenticate your license in two different computers as a maximum, for example your desktop and your laptop. If you try to authenticate the license in a third computer an error message will show up.

How to Create a Calculation

Structural Engineering is all about calculations, isn't it? ***ASDIP Steel*** is a software that allows you perform calculations... and manage them. To create a calculation simply click on any of the calculation buttons in the Project Manager. The dialog box shown below will show up.



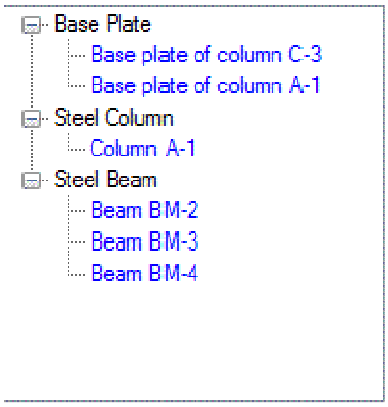


Enter the calculation name or a short, meaningful description of the calculation. When you click the *Add* button the calculation tree at the right half of the Project Manager reflects the change by adding a new node under the corresponding calculation branch. You may continue adding calculations to the tree, or you may open the calculation that you just created.

Think of this as a paper folder on your desk, where you are putting your hand calcs and sketches together. Imagine that you are designing a spread footing by hand and that you ended up with some pages of calcs. If you are a little bit organized, you will put these sheets in a paper folder labeled with the project name. Then imagine that you continue designing by hand another footing of the project, and several more. Add all these paper sheets to your binder. If there are several types of footings in the project, your binder will have to be organized further.

ASDIP Steel organizes the set of calculations for you in an electronic version of the situation described above. The Project Manager will add individual calculations to the project and will organize them in an expandable tree view, similar to the Windows Explorer tree view. You may add as many calculations as you need, and they will be organized properly for future reference.

How to Run a Calculation

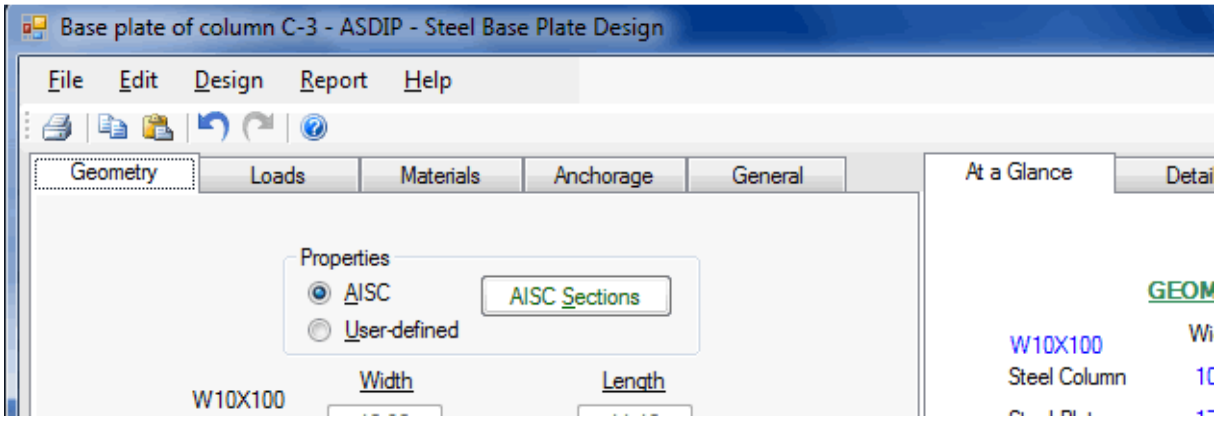


Once you have created a calculation, you can run it by double-clicking on the corresponding node at the tree view in the Project Manager. Alternatively, click on the node to highlight it and then click on the *Open* button. This will retrieve the calculation module. Once there, you may enter the input data and see the results. When you close the calculation sheet, you will be sent back to the Project Manager, which will have the latest information about your calculation, with all the changes that you have just done.

If you have multiple calculations in your project, you can jump to any of them and edit it as needed by simply double-clicking the corresponding node in the tree, as explained above. When you save your project all the information about all your calculations will be saved as well. The next time that you open that project all the calcs will be there.

How to Enter the Input Data

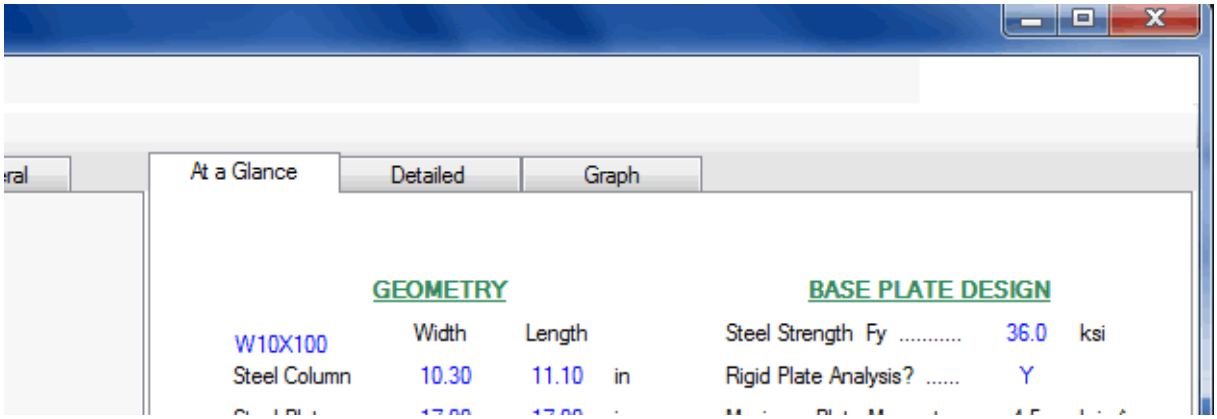
When you open a calculation, a predefined module template is retrieved where you may enter your input data and see immediately the results associated with the change that you have just done. All the modules in ***ASDIP Steel*** have two tabbed controls, as shown below. The left tabbed box is dedicated to the input data, whereas the right tabbed box shows the tabs associated with the results. To enter input data simply select the corresponding tab at the left side and use the text boxes and controls designed for that purpose.



ASDIP Steel has been designed with multiple types of controls in order to enter data the easiest way. Among the controls provided we have buttons, text boxes, labels, combo boxes, ratio buttons, check boxes, tabs, tree views, dialog boxes, menu bars and tool bars. All these combined elements create a rich user interface.

How to See the Results

In ***ASDIP Steel*** the results are always current, which means that as soon as you change any of the input data fields the results are updated accordingly. To see the results, simply click on any of the tabs located at the right half of the screen, as shown below.



The three tabs are described as follows:

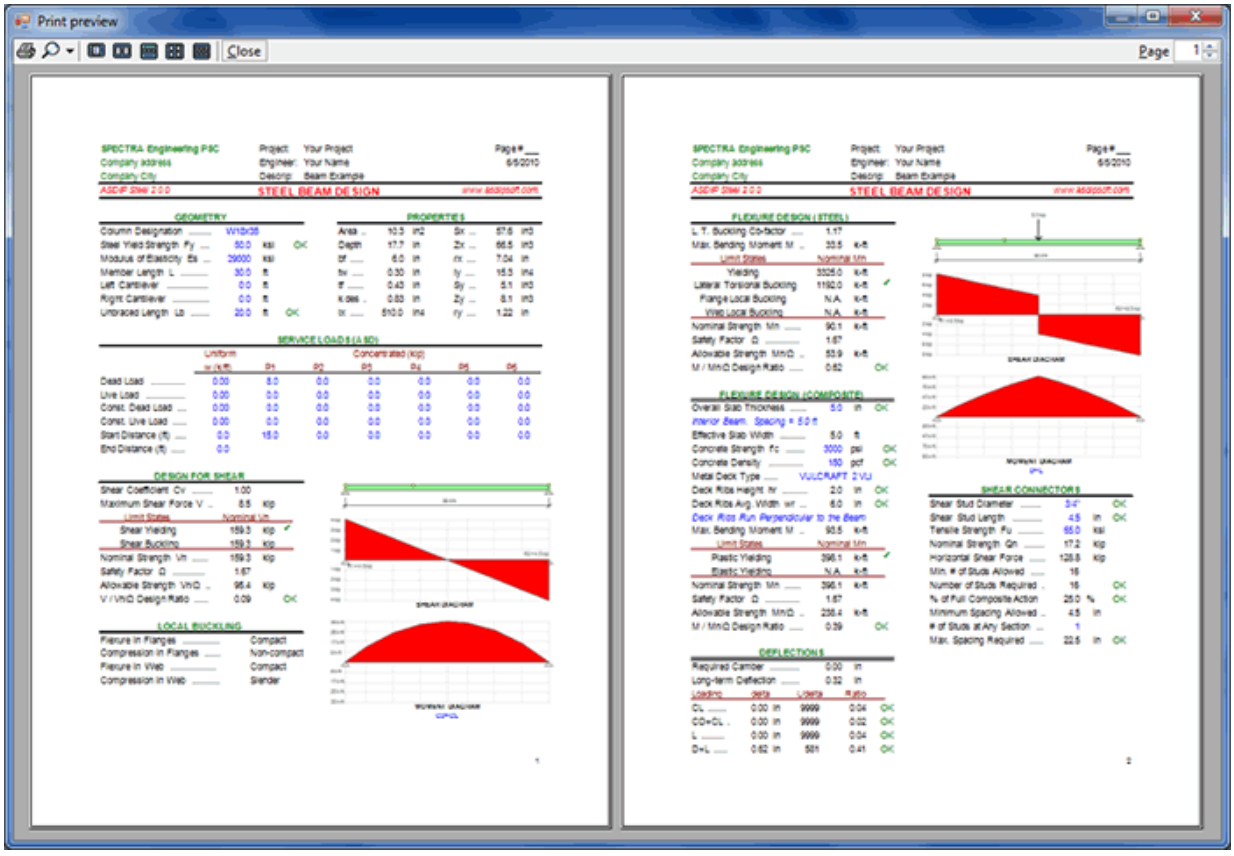
- *At A Glance* tab - Shows a summary of the results, with the most relevant information that fits in one screen height. No scroll bars. The user can see "at a glance" if the design pases or fails. If necessary, you may prefer to see the Detailed tab.
- *Detailed* tab - Shows a more detailed report of the calculations, with more in-depth results. It is intended for a more laborious checking of the step-by-step calculations. The results are organized by

topic for easier reading, and consist of values and text messages always up-to-date. Individual checks show either **OK** or **NG** for pass or fail respectively.

- *Graph* tab - Shows a graphic image related to the calculation, either a plan, elevation, detail, or diagram.

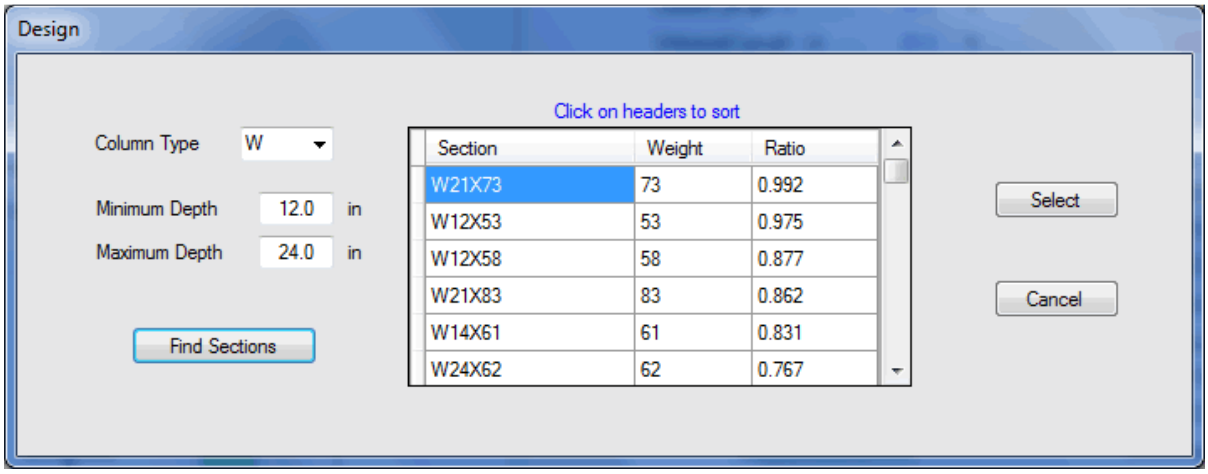
How to Preview the Report

ASDIP Steel generates a pre-formatted report that can be printed out. To preview the report simply select Report | Print Preview from the menu bar. To printout the report, select Report | Print from the menu bar. The preview window provides several buttons to multi-page views, zoom and print, as shown below.



How to Design a Member

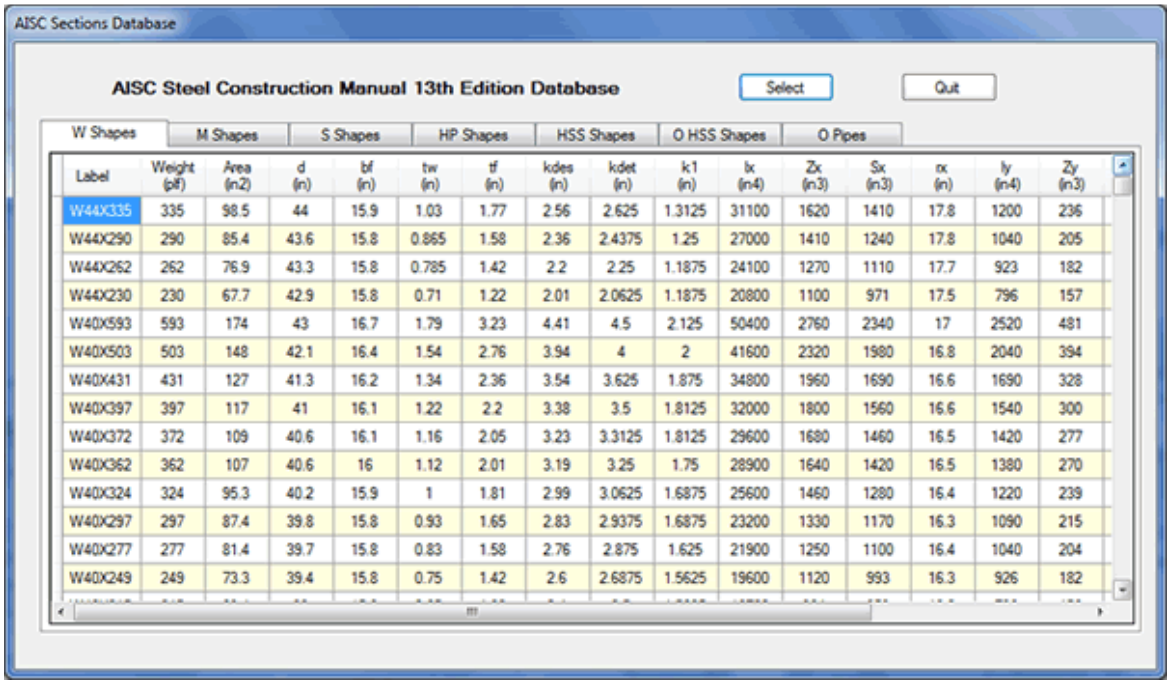
The structural design process is repetitive by nature, since it involves the multiple execution of an algorithm until the solution converges under accepted limits. ***ASDIP Steel*** makes this process straightforward and provides the necessary tools to optimize the design. The image below shows the Design dialog box that shows up when you select Design | Column or Design | Beam from the menu bar.



In this dialog box you may specify the section type and the minimum and maximum section depth. When you click on the *Find Sections* button, the program will run de calculation with all the sections available that meet the specified criteria, and will fill the table with the results with ratio 1.0 or less. The table includes the Section, the Weight and the Ratio and you may sort the table results by clicking on the corresponding header. A ratio of 0.99 indicates that the section is 99% of its capacity. You may select the section that suits your needs from the table by clicking on the *Select* button. The program then automatically shows the module template with the selected steel section.

How to Access the Steel Database

ASDIP Steel has a built-in steel sections database per the AISC Manual 13th Edition, as shown below.

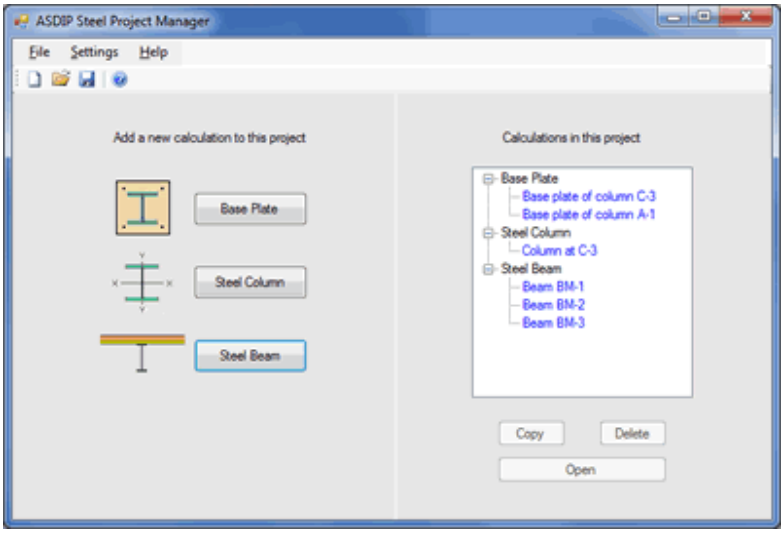


Label	Weight (plf)	Area (in2)	d (in)	bf (in)	tw (in)	tf (in)	kdes (in)	kdet (in)	k1 (in)	Ix (in4)	Zx (in3)	Sx (in3)	rx (in)	Iy (in4)	Zy (in3)
W44X335	335	98.5	44	15.9	1.03	1.77	2.56	2.625	1.3125	31100	1620	1410	17.8	1200	236
W44X290	290	85.4	43.6	15.8	0.865	1.58	2.36	2.4375	1.25	27000	1410	1240	17.8	1040	205
W44X262	262	76.9	43.3	15.8	0.785	1.42	2.2	2.25	1.1875	24100	1270	1110	17.7	923	182
W44X230	230	67.7	42.9	15.8	0.71	1.22	2.01	2.0625	1.1875	20800	1100	971	17.5	796	157
W40X593	593	174	43	16.7	1.79	3.23	4.41	4.5	2.125	50400	2760	2340	17	2520	481
W40X503	503	148	42.1	16.4	1.54	2.76	3.94	4	2	41600	2320	1980	16.8	2040	394
W40X431	431	127	41.3	16.2	1.34	2.36	3.54	3.625	1.875	34800	1960	1690	16.6	1690	328
W40X397	397	117	41	16.1	1.22	2.2	3.38	3.5	1.8125	32000	1800	1560	16.6	1540	300
W40X372	372	109	40.6	16.1	1.16	2.05	3.23	3.3125	1.8125	29600	1680	1460	16.5	1420	277
W40X362	362	107	40.6	16	1.12	2.01	3.19	3.25	1.75	28900	1640	1420	16.5	1380	270
W40X324	324	95.3	40.2	15.9	1	1.81	2.99	3.0625	1.6875	25600	1460	1280	16.4	1220	239
W40X297	297	87.4	39.8	15.8	0.93	1.65	2.83	2.9375	1.6875	23200	1330	1170	16.3	1090	215
W40X277	277	81.4	39.7	15.8	0.83	1.58	2.76	2.875	1.625	21900	1250	1100	16.4	1040	204
W40X249	249	73.3	39.4	15.8	0.75	1.42	2.6	2.6875	1.5625	19600	1120	993	16.3	926	182

To select, click on the tab of the corresponding section type, and then scroll down to find the desired section. To sort, click on the headers. When you click on the Select button, the properties of the selected section are automatically transferred to the calling module.

Project Manager Main Window

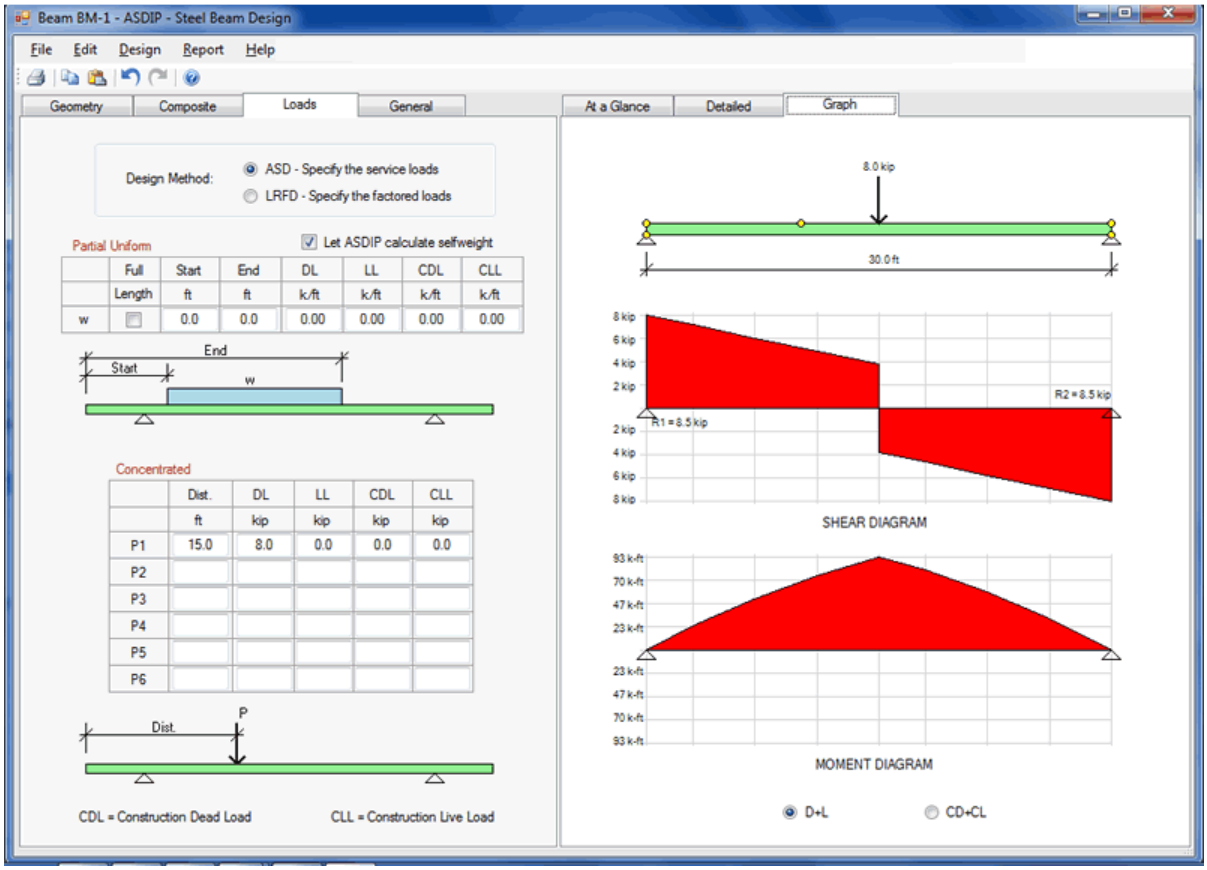
The Project Manager is the central piece of ***ASDIP Steel***. From there all file activities may be performed, as well as get access to the calculation sheets. In the Project Manager you may create a calculation, save a project, set the units system, enter the information and preferences, and organize your work.



Basically the Project Manager is in charge of organizing the calculations in a project, and keeping the common information updated and active. All basic file operations, such as Start a new project, Open an existing project, and Save a project are only possible in the Project Manager. In addition, the expandable tree calculation view is a great way to visualize all the calculations in a project.

Calculation Main Window

Being ASDIP Steel a structural design package, the calculation modules are the essence of the software. All the modules have been designed and developed with a deep understanding of the needs of structural engineers. A typical calculation main window is shown below.



A typical calculation main window is basically composed of the menu bar at the top, and immediately below is the tool bar. Below that is the work area, which consists of two tabbed panes: the left pane is designed to enter the input data, and the right pane shows the results of the calculation. The different tabs have a specific purpose, depending of the calculation module. The current version of ASDIP Steel includes the following calculation modules:

- Steel Base Plate Design
- Steel Column Design
- Steel and Composite Beam Design

Steel Base Plate – Overview

Base plates are elements required at the end of columns to distribute the concentrated load of the column over a much larger area of the material that supports it. The design of column base plates involves two main considerations: One, spread the load so as to maintain the bearing pressures under the allowable values, and the second is with the connection, or anchorage, of the base plate and column to the concrete Steel.

For frame analysis it may be important to consider the fixity of the column base, so the design of the base plate and the anchorage to the Steel should consider the interaction of the axial load and bending moment.

The module performs the elastic design of a column steel base plate resting on a concrete pier and subjected to any combination of axial load and bending moment, including uplift loading. The moment is assumed acting about the strong axis of a steel column welded to the plate. In addition, this program computes and checks the maximum bearing stress on the pier, as well as the tension and shear forces per rod, necessary to design the anchorage system.

The program designs the anchorage per the latest provisions of the ACI, and includes checks for all failure modes in tension and shear, as well as anchor reinforcing design.

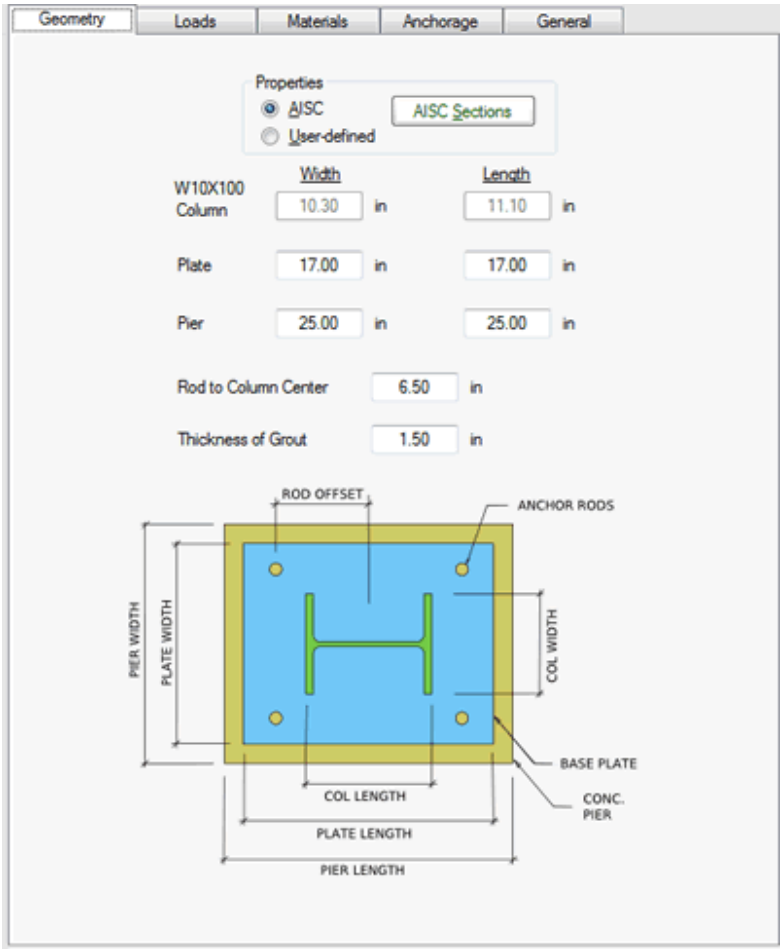
For axially loaded base plates, such as those in frames assumed to be pinned at the base, the program is based on either the cantilever model or the Thornton method covered in the AISC Manual 13th edition.

For base plates with moment, two design theories are considered:

- For plates assumed rigid, the strain compatibility is enforced in accordance with the Blodgett method ("Design of Welded Structures").
- For plates assumed flexible, the strain compatibility is ignored in accordance with the DeWolf method ("AISC.Design Guides # 1, Second Edition").

Steel Base Plate – Geometry

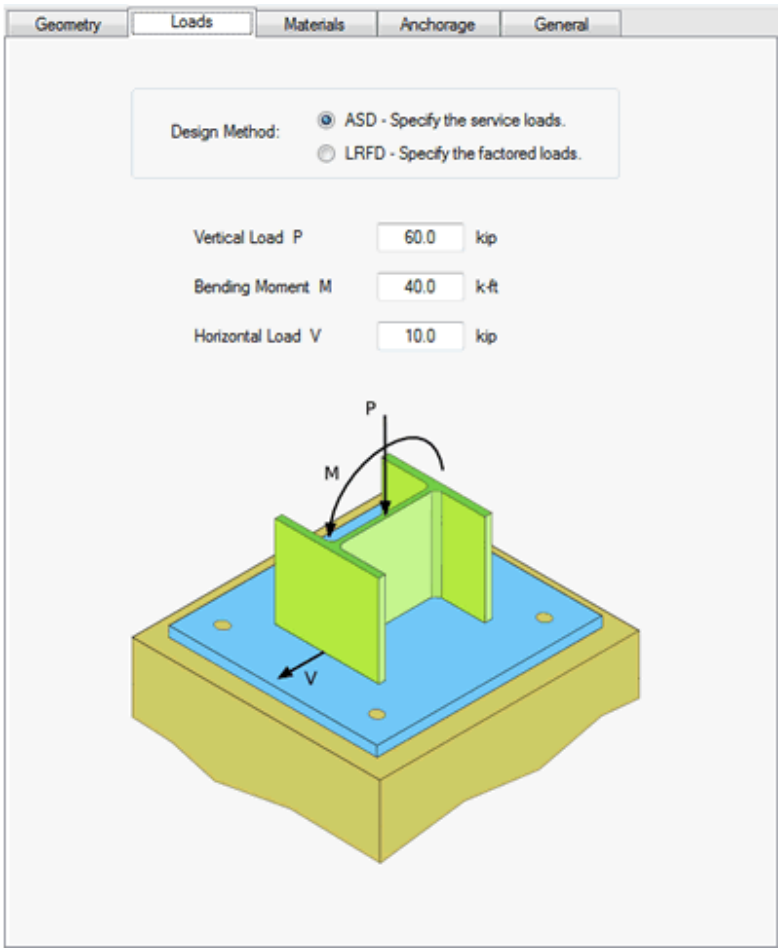
Use the *Geometry* tab to enter the information of the dimensions of the column base, as shown below.



At the top of the page, the *Column Properties* section allows you specify the steel column. You may select either AISC or User-defined. The former enables the *AISC Sections* button, which will show the built-in steel sections database per the 13th edition AISC Manual. Selecting a section from the database will bring the properties into the *Geometry* tab automatically. User-defined allows the user to enter the column dimensions directly.

Steel Base Plate – Loads

The *Loads* tab lets you enter the loads transferred from the column, as shown below.

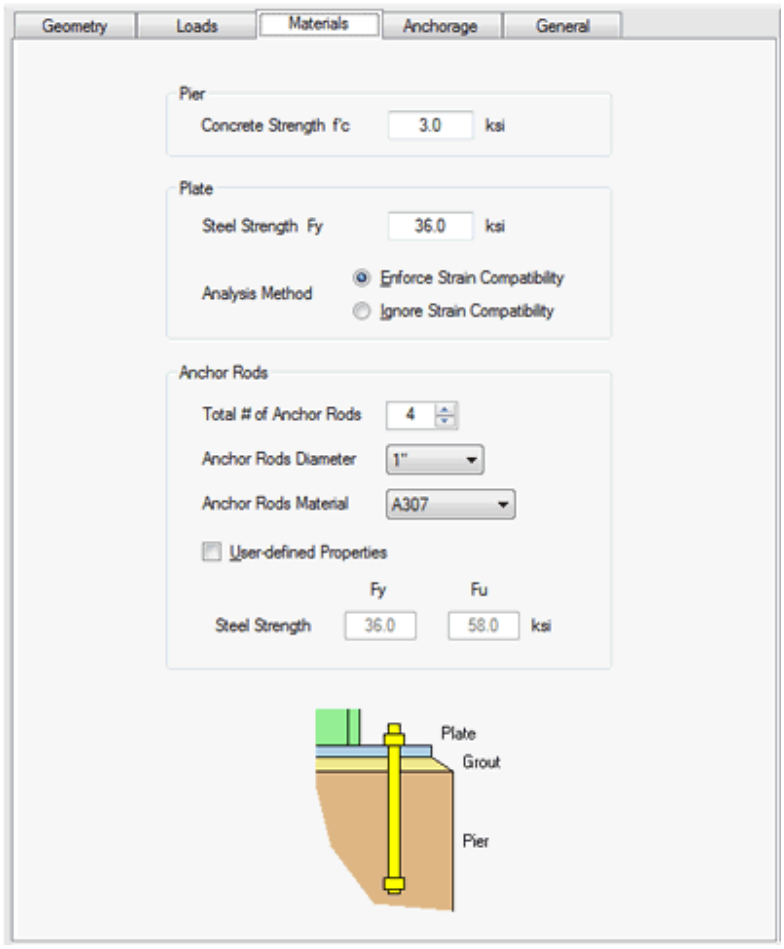


ASDP Steel allows you to specify the design method based either on the ASD or LRFD methodology of the AISC. For ASD enter the service loads. For LRFD enter the factored loads.

The loads representing the column reactions are the vertical load, the bending moment, and the horizontal load. For axial tension or uplift enter the vertical load as a negative value.

Steel Base Plate – Materials

The *Materials* tab is designed to enter the information about the pier, the plate, and the anchor rods, as shown below.



ASDIP Steel performs the plate design based on two different theories:

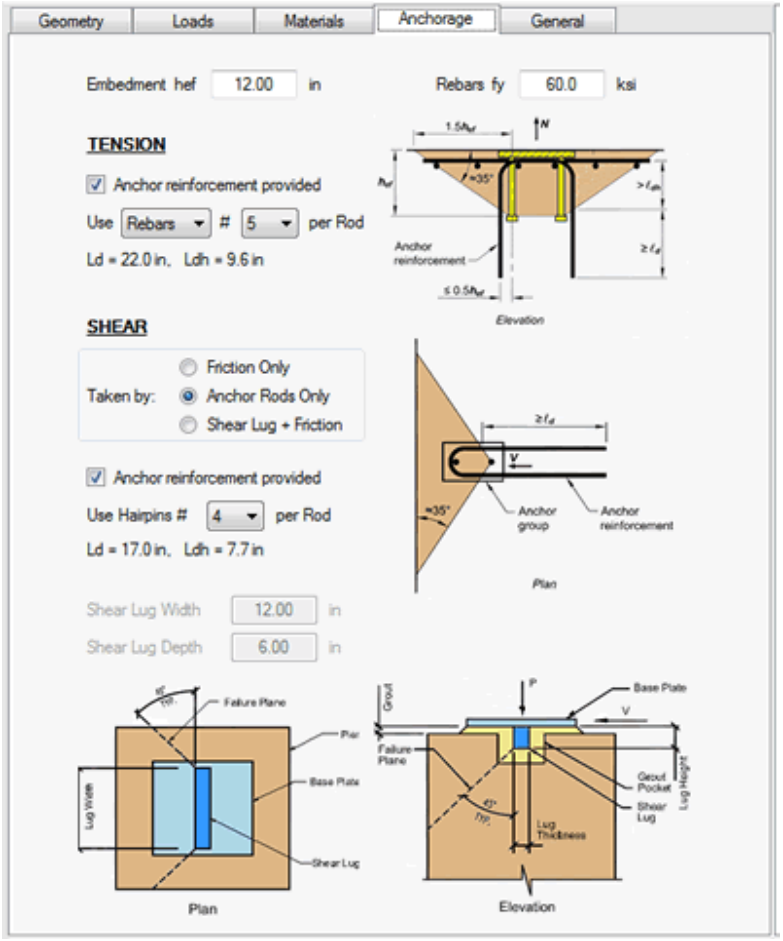
- Strain compatibility is enforced in accordance with the Blodgett method ("Design of Welded Structures")
- Strain compatibility is ignored in accordance with the DeWolf method ("AISC.Design Guides # 1, Second Edition")

The material of the anchor rods may be specified two ways:

- Select the material from the pull-down list
- Mark the User-defined check box, and enter the properties directly.

Steel Base Plate – Anchorage

The *Anchorage* tab has been included to enter all the required information to design the anchorage of the base plate to the Steel, as shown below.



ASDIP Steel fully complies with the provisions of the ACI-318 Appendix D "Anchoring to Concrete". The anchor rods are designed considering all the failure modes in both tension and shear, and the interaction effects are checked.

For tension, ***ASDIP Steel*** checks the steel failure, pullout, concrete breakout, and side-face blowout. In addition, it designs the anchor reinforcement in order to transfer the anchor force to the Steel.

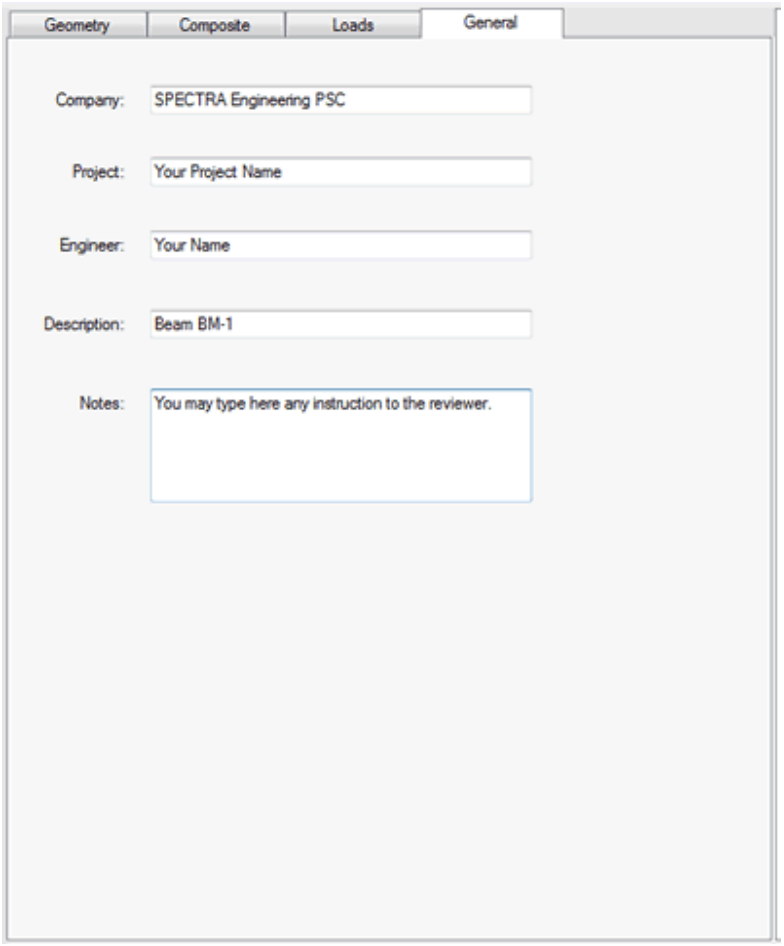
For shear, there are three alternatives to take the load:

- Friction Only - With this option the shear is entirely taken by the friction between the base plate and the concrete. No shear force is transferred to the anchor rods.
- Anchor Rods Only - Friction is neglected, and the force is taken by anchor rods located at the front face only. This is to account for the fact that it's unlikely that all the rods will be in contact with the plate. ***ASDIP Steel*** checks the steel failure, concrete pryout, and concrete breakout. In addition, it designs the anchor reinforcement in order to transfer the anchor force to the Steel.

- Shear Lug + Friction - The shear is taken by friction, and the remaining is taken by a shear lug. ***ASDIP Steel*** will check the bearing stress at the lug and design the plate thickness accordingly.

General tab

The *General* tab contains information about the calculation that may be of interest to the designer or to someone else who reviews the calculation, as shown below.



Company - The company name was setup when you installed the license and cannot be changed.
Project - Since the project name is common for all the calculations of this project, it can only be specified in the Settings | Information menu command of the Project Manager.
Engineer - When you create a calculation, the engineer's name is brought from the user information that you entered in the Project Manager. Since two calculations of the same project may be done by two different engineers, this text can be changed in your calculation.
Description - By default, this is the calculation name that you entered when you created the calculation. This text, however, can be changed to a more meaningful description if desired. This text will be copied into the report.

Notes - This text is intended to provide any information to the reviewer, or a remainder to yourself in the future.

Steel Column – Overview

Columns are structural members that mostly work in compression and bending, and only very short columns can be axially loaded to their yield stress. Often buckling, or sudden bending as a result of instability, occurs prior to developing the full material strength.

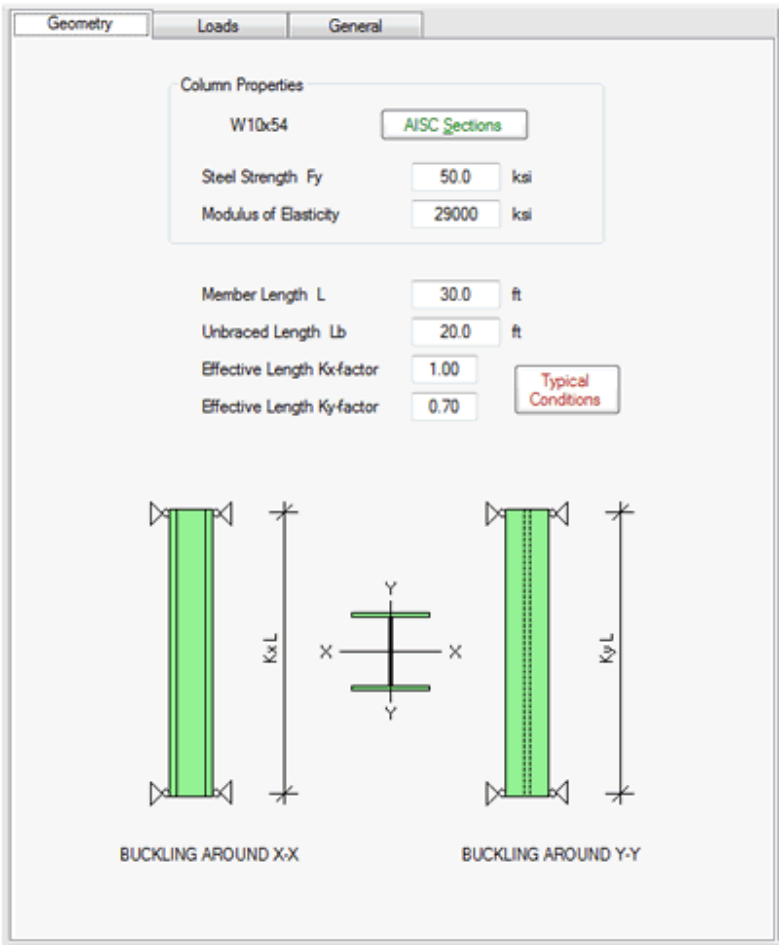
The program performs the design of a steel member subjected to axial load and bending moments about its two principal axes. The program is based on the AISC ASD/LRFD methodology and checks the axial, bending, and combined stresses according to the AISC 360 Specifications (13th Ed. Manual). Either service or factored loads may be specified.

Two types of procedures may be followed in order to calculate the required strength and the design requirements, depending on the source of the loads entered as data:

- Second Order Analysis, which considers the P-Delta and P-delta effects.
- Amplified First Order Analysis. In this case the program will calculate the amplified moments based on the information provided.

Steel Column – Geometry

The *Geometry* tab allows you to enter the dimensions and properties of the column, as shown below.

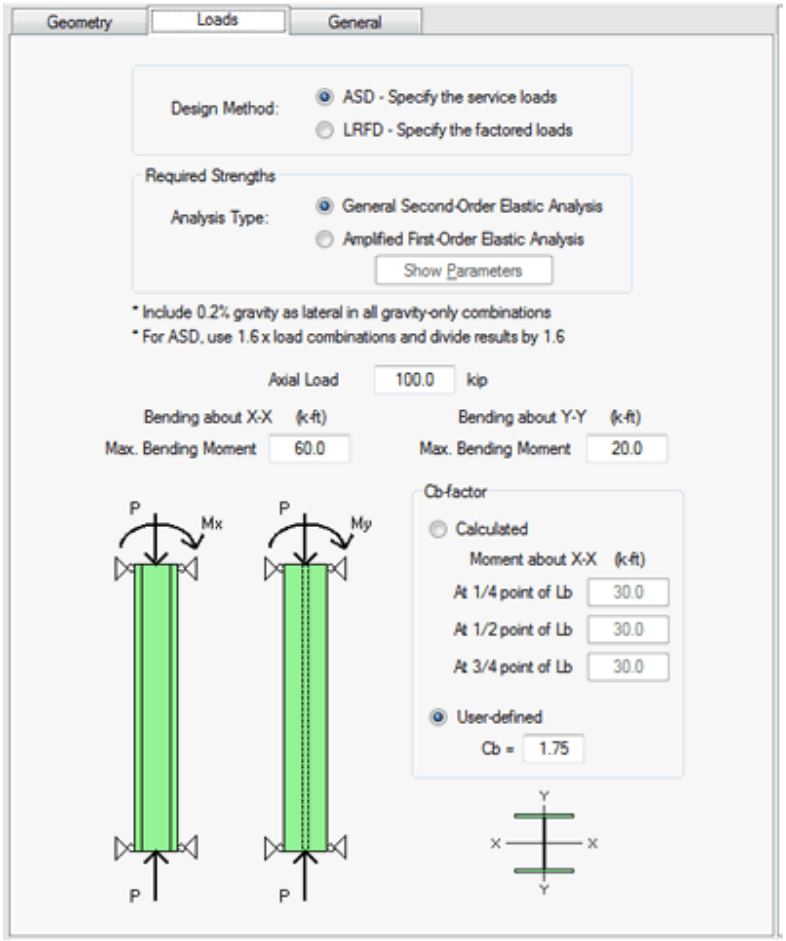


Click on the *AISC Sections* button to invoke the built-in steel sections database per the 13th edition AISC Manual. Selecting a section from the database will bring the properties into the *Geometry* tab automatically.

The Typical Conditions button will show up the AISC recommended values for the effective length K factor when ideal conditions are approximated.

Steel Column – Loads

ASDP Steel allows you to specify the design method based either on the ASD or LRFD methodology of the AISC. For ASD enter the service loads. For LRFD enter the factored loads.



Two types of procedures may be followed in order to calculate the required strength and the design requirements, depending on the source of the loads entered as data:

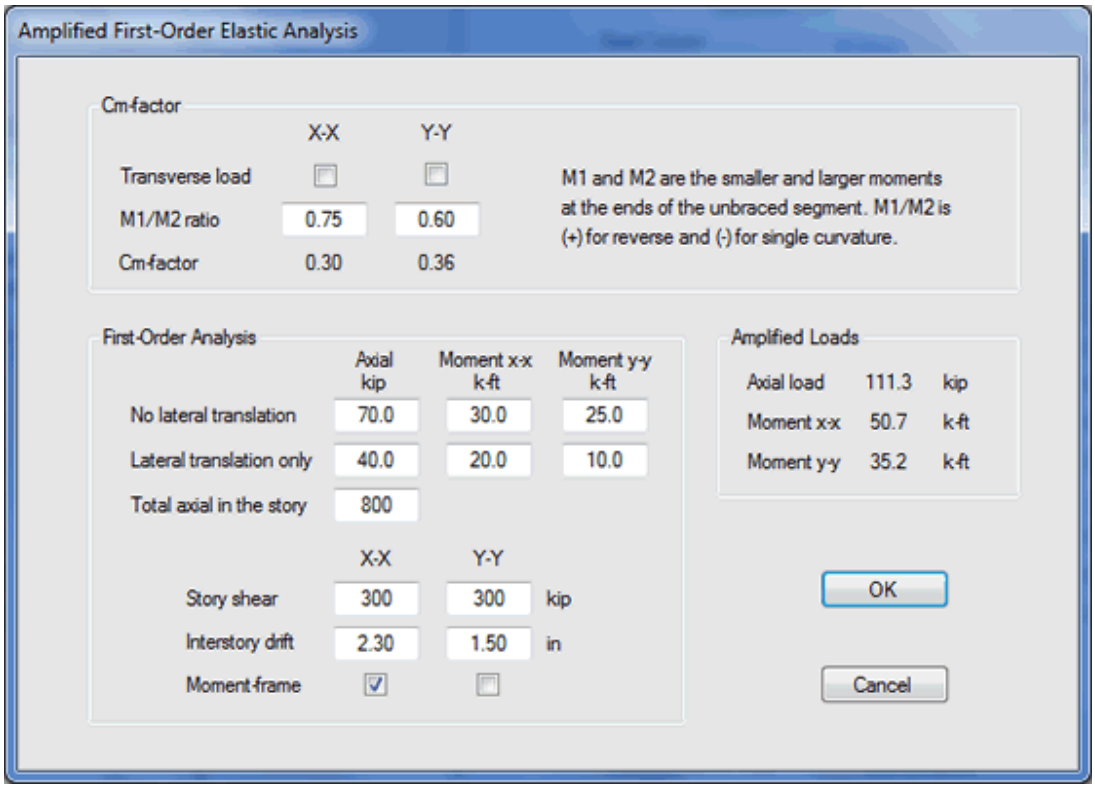
- Second Order Analysis, which considers the P-Delta and P-delta effects.
- Amplified First Order Analysis. In this case the program will calculate the amplified moments based on the information provided.

The program performs the design of a steel member subjected to axial load and bending moments about its two principal axes. The lateral torsional buckling Cb factor may be either calculated or user-defined.

Amplified First-Order Analysis

According to AISC C2, required strengths shall be determined using a Second-order Analysis that considers both P-Delta and P-delta effects. An accepted alternate method is the Amplified First-order Analysis for braced, moment, and combined framing systems.

ASDIP Steel allows you to specify the analysis type. When the Second-order Analysis is selected, the loads that you enter are used directly as the required strengths. When the Amplified First-order Analysis is chosen, the entered loads are assumed to come from a first-order analysis and the program internally amplifies the axial forces and moments per the procedure in AISC C2.1b, as shown below. These amplified forces are then used as the required strengths.



Steel Beam – Overview

Beams are structural members that mostly work in bending and shear as a result of transverse loading. Other terms such as girders, joists, purlins, stringers, girts and lintels are often used. The compression flange, which is attached to the web in the plane of the beam, may or may not be laterally braced, thus the buckling concepts of compression members apply to beams as well.

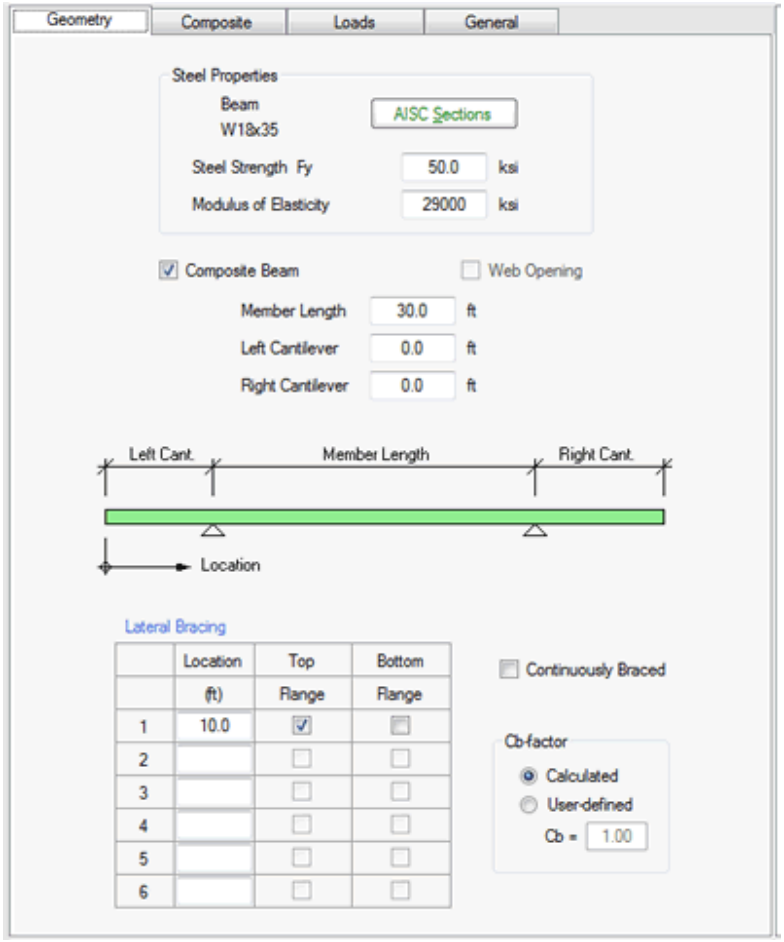
Composite action is developed when two load-carrying structural elements, such as a concrete floor slab and its supporting steel beams, are integrally connected and deflect as a single unit. The stiffness of a composite floor is substantially greater than that of a concrete floor and its supporting beams acting independently. In addition, a 20 to 30% savings in steel weight is often possible by taking full advantage of a composite system. Since the concrete slab exists anyway and the shear connectors are inexpensive and easy to install, it is structurally advisable to use composite construction whenever possible.

The program performs the design of a simply supported, either interior or edge, steel or composite beam subjected to distributed and concentrated loads. Two cantilevers may be modeled. The program computes the maximum bending moment, shear force, and vertical deflection induced by the applied loads, and compares them against the beam strength. The program computes the number and spacing of the shear connectors required to develop either partial or full composite action, as well as the required camber.

The module is based on the AISC ASD/LRFD methodology and calculates the shear and flexure strengths according to the AISC 360-05 Specifications (13th Ed. Manual). Either service or factored loads may be specified.

Steel Beam – Geometry

The *Geometry* tab allows you to enter the dimensions, properties, and lateral bracing of the beam, as shown below.



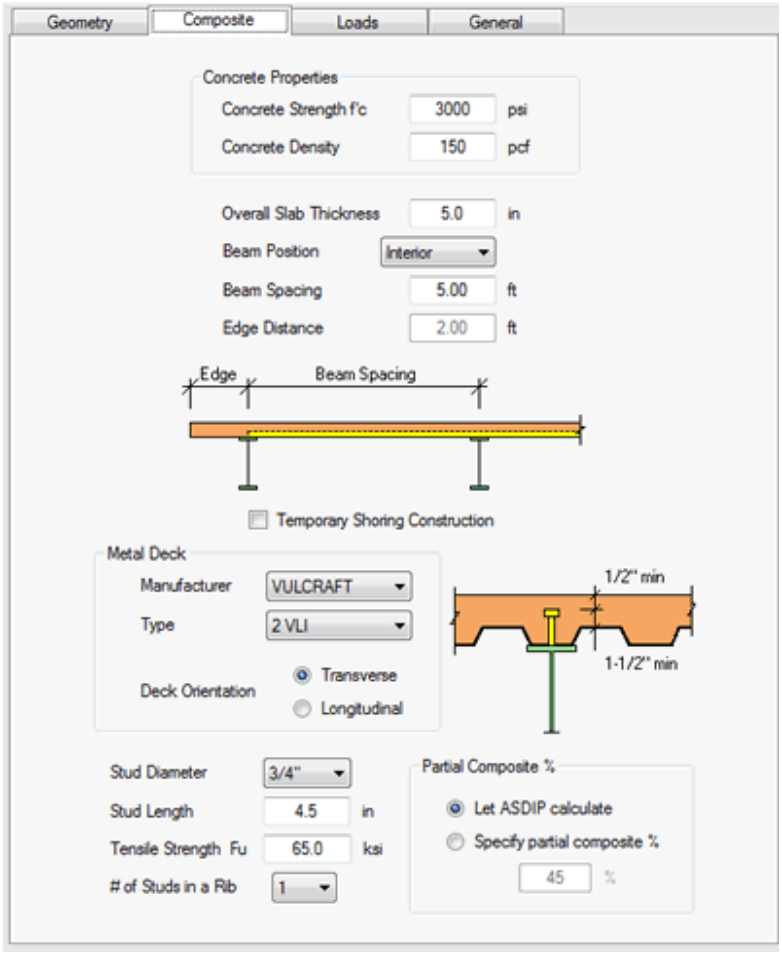
Click on the *AISC Sections* button to invoke the built-in steel sections database per the 13th edition AISC Manual. Selecting a section from the database will bring the properties into the *Geometry* tab automatically.

If the beam is composite, check mark the *Composite Beam* check box. This will add the *Composite* tab to the module. Un-checking this box will remove this tab.

ASDIP Steel allows you to specify the location of the lateral bracing, defined in AISC F2.2 as "points that are either braced against lateral displacement of compression flange or braced against twist of the cross section". The program internally will calculate if the braced flange is in compression, otherwise the bracing at that point will be ignored. The lateral bracing applies also to composite beams, since during the construction stage the beam is not composite.

Steel Beam – Composite

ASDIP Steel will design either steel (non-composite) or composite beams. If the beam is composite, check mark the *Composite Beam* check box in the *Geometry* tab. This will add the *Composite* tab to the module, as shown below.



In this tab you may specify all the characteristics of the composite section, such as the concrete properties and the slab thickness, type of deck, and shear connectors. The structural properties of the composite section depend of the beam position: either Interior or Edge. For an edge beam, the edge distance is required.

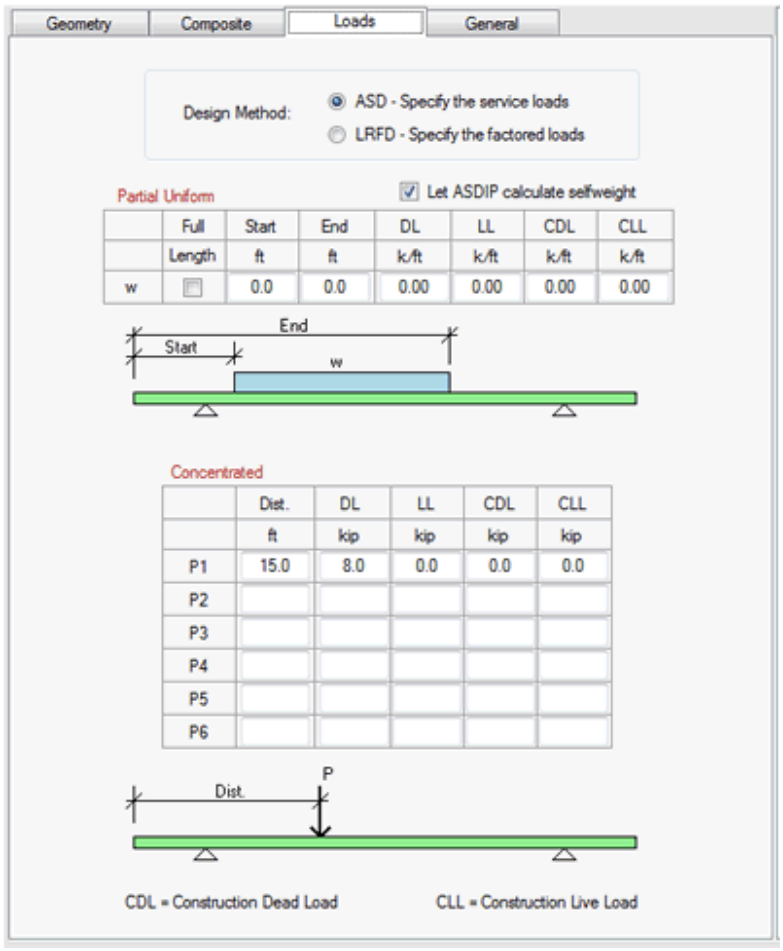
If the *Temporary Shoring Construction* check box is marked, it will indicate that the non-composite steel beam during construction doesn't need to be checked. On the other hand, if the beam is not shored, ***ASDIP Steel*** will perform the design of the non-composite steel beam under the construction dead and live loads.

The metal deck is an important element in a composite beam, since it will affect the structural properties of the composite section. ***ASDIP Steel*** has a built-in database of deck manufacturers and the corresponding deck types that can be selected from the pull down list. Products from other manufacturers are essentially similar.

The shear studs are specified at the bottom section of the tab. ***ASDIP Steel*** will internally check the minimum dimensions represented on the figure. In addition, partial composite action can be specified easily, either by letting ASDIP calculate the required studs or by specifying the partial %.

Steel Beam – Loads

The *Loads* tab has been designed to enter all the information related to the applied loads to the beam, as shown below.



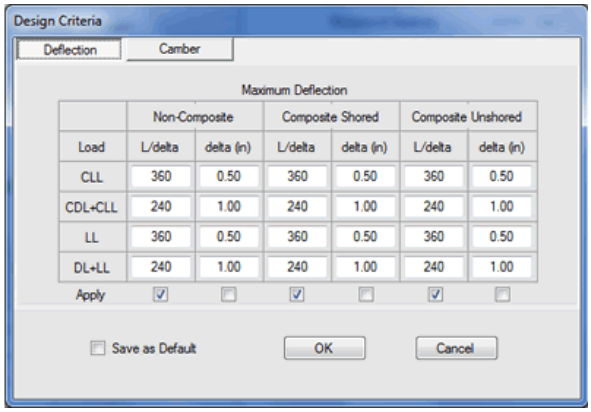
ASDIP Steel fully complies with the AISC 360-05 specifications (13th Ed. Manual), which states in B3 that designs shall be made according to the provisions for the LRFD or the provisions of the ASD. You may specify the type of load at the top of the tab.

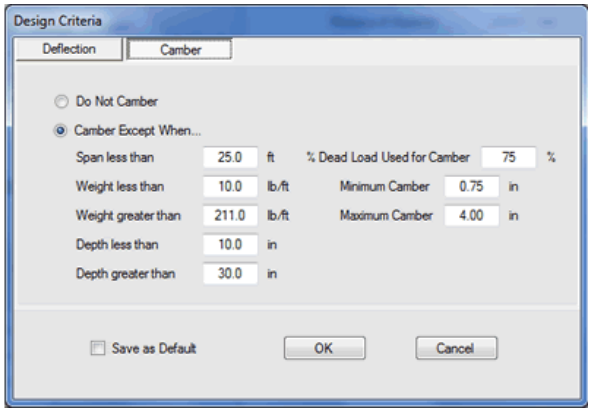
ASDIP Steel allows you to specify one partial uniform load and up to six concentrated loads. For an internal calculation of the selfweight, check mark the *Let ASDIP calculate selfweight* check box. In all instances, you may specify the following load cases for either ASD or LRFD:

- Dead Load (DL) - Dead load after the concrete has reached 75% of its strength.
- Live Load (LL) - Live load after the concrete has reached 75% of its strength.
- Construction Dead Load (CDL) - Dead load before the concrete has reached 75% of its strength.
- Construction Live Load (CLL) - Live load before the concrete has reached 75% of its strength.

Steel Beam - Design Criteria

The Design Criteria dialog box may be invoked by selecting Design | Criteria from the menu bar, and specifies the requirements and limitations that the selected section has to comply with. It has two tabs, one for Deflection, and one for Camber criteria, as shown below.





The *Deflection* tab sets the maximum allowable deflection in the beam for all the loading conditions, and it can be specified as an absolute delta, as a relative L/delta, or both.

The *Camber* tab sets the criteria to calculate the camber in the beam, as well as the maximum and minimum camber.

File Menu

New
Open
Save
Save As
Go to Project Manager
Exit ASDIP Steel

File | New

This command is only available in the Project Manager and it will clear all the information from previous projects, such as the user and project information and calculations. Remember to save your work before you start a new project, otherwise it will be lost. ***ASDIP Steel*** will ask you to confirm, in case that you haven't saved your previous project.

File | Open

This command is only available in the Project Manager and it will retrieve all the information contained in a file previously saved in ***ASDIP Steel***. The Open Project dialog box will pop up, so that you may specify where the file is located.

File | Save

This command is only available in the Project Manager and it will save all the information regarding your project in a file with extension .sdp, which is the default extension for ***ASDIP Steel*** projects. If the project has been saved previously, it will be saved directly, otherwise the Save Project dialog box will pop up, so that you can specify the location of the file.

File | Save As

This command is only available in the Project Manager and it will save all the information regarding your project in a file with extension .sdp, which is the default extension for ***ASDIP Steel*** projects. The Save Project dialog box will pop up, so that you can specify the name and location of the file.

File | Go to Project Manager

This command is only available in the Calculation modules and it will close the current calculation sheet and will take you back to the Project Manager. From there, you may either open another calculation or save your work. It has the same effect as clicking on the X at the upper right corner of the calculation. Note that all your input data and results are not lost, they are still in memory. If you open your calculation again you will see all the information there.

File | Exit ASDIP Steel

This option will close and terminate the application. ***ASDIP Steel*** will ask you to confirm, in case that you haven't saved your project. If you accept, the program will close immediately.

Settings Menu

Units – Sets the system of units of the whole project.
Information – Enter basic information of both the user and the project.
Preferences – Sets different options to customize your experience with ***ASDIP Steel***.
License – Lets you authenticate your license.

Edit Menu

Undo - It will reverse the last command
Redo - It will reverse the last undo operation
Copy - It will store the highlighted text in the clipboard
Paste - It will place the clipboard contents in the current position

Design Menu

Units – Sets the system of units for the current calculation, otherwise it uses the default .
Criteria – Sets the design parameters to be used in the calculation.
Beam – Lets you design your beam according to the specified criteria.

Report Menu

Print - It will show the *Print* dialog box
Print Preview

Help Menu

Contents - Shows the Table of Contents of the Help file, where you may select by topic.
Index - Shows the Index tab, where you may select by keyword.
Search - Shows the Search tab, where you may find a word in the Help file.
About - Shows the ***ASDIP Steel*** information dialog.