Section 21
Computer Programs

21.1 Guidelines on Use

This section provides an overview of the software applications currently in use by NYSDOT. It should not be construed as an endorsement of any particular software by NYSDOT. Unless noted by contract, consultants to NYSDOT are not required to use the software listed in this section. Users should refer to the corresponding manuals for more detailed instructions, specifications, and limitations.

Computer programs have become valuable tools for the engineer by automating repetitive design calculations. Even the best programs, however, will not give correct results if the input is not correct. Therefore, any computer program used shall be checked with a second program or enough hand calculations to verify results. Program input shall always be checked by a second designer. All input and results shall be printed out and placed in the design folder.

Users in the Department should also visit the Engineering Programs Support (EPS) internal web site for information on bridge design software that is used in the Main Office and Regions. The site can be accessed from the Office of Structures Home Page by clicking on the Design Information tab then selecting Engineering Program Support. Here you will find the latest information on the various programs, including manuals, tutorials and other important documents.

21.2 Hydraulics Programs

The following programs are available to determine the flood intensity, water elevations, scour potential, check FEMA compliance and help size the proposed structures over waterways.

- **BRI-STARS** is a pseudo two-dimensional hydraulic program that (through the use of stream tubes) provides a time and flow dependent two-dimensional sediment routing (aggradation and degradation) model in a bridge cross section.

- **HEC-1** and **HEC-HMS** developed by the Corps of Engineers to provide watershed runoff and routing capabilities.

- **HEC-GeoHMS** is a tool developed by the Corps of Engineers to translate GIS spatial information into hydrologic models. It is an extension to ArcView GIS that processes Digital Elevation Models (DEM) to determine drainage basin delineation, areas, flow paths, elevation and other hydrologic parameters. NYSDOT uses these data in the USGS regression equations to determine stream peak flow.

- **HEC-RAS** is the updated version of the HEC-2 computer program developed by the Corps of Engineers which computes the water surface profiles and velocities using the stream cross sections, Manning's roughness and input flows. This program can handle variable
flows and has a WSPRO subroutine in their water profile routine. In addition, this program computes the possible scour depths at the substructures. HEC-RAS replaces the obsolete HEC-2 program. HEC-2 was used by FEMA for flood plain studies prior to the development of HEC-RAS.

- **HYDRAIN** is a collection of several programs developed as a pool fund project by several states and FHWA. It includes HYDRO a hydrology program; HYDRA which simulates hydrology and hydraulics on storm drain or sanitary pipes network; HY8 which simulates hydraulic analysis or design for culverts, reservoir routing and energy dissipaters; HYCHL which analyses and designs channel and rip-rap linings; and NFF which interactively calculates USGS Regression equation flows. It is no longer available as a stand-alone, but its functions are included in WMS.

- **HydrologyAdj.qpw** is an in-house developed Hydrology program that computes the flows for the USGS Regression equations (similar to the NFF subroutine equations in HYDRAIN), and adjusts per USGS WRI 90-4197 if there is a nearby gage on the stream.

- **Scour Analysis Spreadsheet HEC-18 Evaluating Scour at Bridges, 4th Edition (FHWA NHI 01-001 is the NYSDOT standard reference for analyzing contraction, pier, and abutment scour at bridges. The Hydraulics Unit has developed an Excel spread sheet using the HEC-18 equations to input data, calculate scour parameters, and present the results in organized format.

- **SMS using FESWMS or SMS using RMA-2** the Surface Modeling System creates a graphic interface which uses either FESWMS or RMA-2 to perform 2D hydraulic analysis on complicated hydraulic models that require two dimensional analysis, such as at confluences or multiple-inlet tidal bays where the geometry cannot be adequately modeled with one dimensional model.

- **WMS**, a Watershed Modeling System, is a hydrologic program that uses digital terrain modeling (DEMs and TINs) to delineate drainage basins, and compute drainage basin parameters to develop peak flow estimates and hydrographs. It also includes the functions of the HYDRAIN system, which is no longer available or supported as a stand-alone.

- **WSPRO** is a computer program developed by FHWA which computes the water surface profiles and velocities using the stream cross sections Manning’s variable and the Design ($Q_{50}$) and Basic ($Q_{100}$) flows.

### 21.3 Structures Programs

#### 21.3.1 In-House Programs

The in-house analysis and design programs listed below were developed on the basis of standard AASHTO (ASD and LFD) design. They are no longer updated, and their use will be limited because the department has adopted LRFD as the governing bridge design specifications. The geometry programs will continue to be used.

- **CASH** (Cantilever, Anchor, Sheet Pile, H-Pile Program) designs and analyzes cantilevered, soldier piles with lagging or anchored flexible earth retaining walls. It uses the Blum
(Simplified) Method and the Jumikis (Conventional) Method in the analysis of cantilever walls, and the Free Earth Support Method to analyze anchored flexible walls. The program makes provisions for cohesive or cohesionless soils, resistance reduction factors, soil slopes, ground water, and surcharges.

- **CULVERT** will design and/or analyze a one-, two-, three-, or four-cell reinforced concrete box culvert with prismatic members (precast or cast-in-place) with or without a bottom slab, using either the working stress or load factor method. All cells are assumed to be the same size for any one culvert and the clear opening dimensions remain constant during the design process. By knowing the span, rise, and fill height, the program will design the box culvert by either service load or load factor design. It will display the bar schedule for the entire length of a cast-in-place box culvert or simply one unit of a precast box culvert.

**Mathcad Worksheets.** The following worksheets, available on our website (nysdot.gov), were created in-house:

- LRFD Bearing Design worksheets that were previously published as Appendices to Section 12.
- LRFD Abutment and Retaining Wall worksheets.
- LRFD Gusset Plate Design and Analysis worksheet is for use on new structures. See Structures Design Advisory 08-001 for more information.
- LFD Gusset Plate Analysis and LFR Load Rating worksheet is for use on existing structures. See Structures Technical Advisory 09-001 for more information.

**STRAPPS (STRuctures APPlicationS)** is a group of programs written and maintained by personnel of the Office of Structures of NYSDOT to aid in structural design and analysis. They include PIERRUN, CAPBEAM, CONTFTG, INDVFTG, WALL, SUPGEOM, SCUPPER, SPLICED, VERTCL, and COLUMNU. A Windows interface has been created for the SUPGEOM, SCUPPER, CULVERT, WALLRUN and VERTCL programs.

- **COLUMNU** is a program to design reinforced concrete compression members to resist a given combination of loadings or to investigate the adequacy of a given cross section to resist a similar set of loadings. Each loading case consists of an axial compressive load combined with uniaxial or biaxial bending. The method of solution is based on ultimate strength theories for reinforced concrete design.

- **PIERRUN** (Pier Analysis Program) is a control program for a software suite that also provides data to three other subprograms: CAPBEAM, CONTFTG, and INDVFTG. This suite handles input and will completely analyze and design a reinforced concrete, multiple column, rigid frame bridge pier of up to six columns, or a single-column, hammerhead bridge pier. PIERRUN analyzes the input using an exact method of indeterminate frame analysis and stores the moments, shears, and axial loads resulting from the analysis in a data file.
Input for PIERRUN includes a description of the frame and superstructure geometries, and the magnitudes of the various applied loads, or sufficient information necessary to compute these loads. The frame may consist of one to six columns. The columns may be round or rectangular in cross section, and may be tapered in either direction. The capbeam may consist of interior spans and cantilevers, all of which may be haunched linearly or parabolically. The superstructure which the frame supports may consist of up to 30 stringers positioned anywhere on the pier. Up to ten vehicle or sidewalk lanes may be positioned anywhere on the superstructure. Column fixities at the base may be assigned a value which may vary from pinned to fully rigid. The program assembles the individual loadings into AASHTO group loadings and an analysis of these AASHTO group loadings is performed based on either service load or load factor criteria. The design option for PIERRUN and its subprograms uses working stress theory.

- **CAPBEAM** program uses the data produced by PIERRUN to design the positive and negative longitudinal steel reinforcement in the capbeam, and will design double vertical stirrups for diagonal tension shear.

- **CONTFTG** is a Continuous Footing Design program. It will design the pile pattern in a rectangular grid for pile footings, will determine all footing dimensions, and will design the positive and negative reinforcing steel along the parallel axis and the top and bottom steel along the normal axis. The footing length is determined so that the positive and negative moments are balanced. The pile pattern will be a rectangular grid which results in the minimum number of piles. For spread footings, the width will be the minimum required for the length. The footing depth will be sufficient so that diagonal tension reinforcement is not required.

- **INDVFTG** is an Individual Footing Design program. It will determine the pile pattern for pile footings, the footing dimensions, and will design the top and bottom steel along both axes. For pile footings, the pile pattern will be that which results in the minimum number of piles. For spread footings, the footing area will be a minimum, but in no case will one dimension be larger than twice the other. The footing depth will be sufficient so that diagonal tension reinforcement is not required.

- **SCUPPER** is a program that designs bridge deck and bridge end drainage facilities based on user input describing the rain intensity and the length, slope and cross-section of the structure.

- **SPLICED** (SPLICE Design Program) was developed for the design and review of bolted splices in both plate girders and rolled beams designed to handle loads and stresses induced by highway loadings. Details such as plate and bolt clearances as well as additional plate thickening for corrosion are considered. Use of the program should be coordinated through the Metals Engineering Unit.

- **SUPGEOM** (Bridge SUPERstructure GEOMETRY Program) is a bridge layout program that processes user input to compute the azimuth of each beam, length between working lines, span length, and elevations at a chosen interval. Haunch, camber, offsets along the working line of each end of the beam, and normal or radial offsets of the beam at the designated points from the station line are also computed. In the case of a fascia beam, the overhangs at designated points along the beam will also be computed.
• **VERTCL** (Shoulder Break and **VERTical CLearance Program**) is used to calculate the vertical clearance under a structure, the allowable beam depth, and the shoulder break points of the over roadway for the preliminary layout of a structure over a highway, stream, or railroad. The program’s input data must consist of horizontal and vertical alignment as well as cross-section information on both the over and under roadways.

• **WALL** uses working stress to design or analyze the major elements of a stub abutment, high (or solid) abutment, or a retaining wall. For each type of structure, the program designs stem steel at critical points, footing dimensions, footing steel, and the number of pile rows and pile spacing if piles are used. The type of footing must be predetermined and the permissible soil pressure or pile loads known.

### 21.3.2 Commercial Programs

The following software have been obtained by NYSDOT from commercial providers, and are currently in use by the Department:

• **AISIsplice**
  
  AISIsplice is a program that was used for the analysis and design of bolted field splices for straight, I-shaped steel girders on the basis of AASHTO LRFD specifications. However, this program is no longer available. We are currently working on a Mathcad worksheet for splice design as a replacement.

• **BRADD (BRidge Automated Design and Drafting System)** is a computer software system that was developed for NYSDOT to automate the bridge design and drafting process. BRADD is no longer used for finished designs. It may be used to get preliminary data.

• **BRASS (Bridge Rating and Analysis of Structural Systems)** is a software package consisting of multiple modules capable of designing or analyzing girders (BRASS-GIRDER), piers (BRASS-PIER), culverts (BRASS-CULVERT), trusses (BRASS-TRUSS), splices (BRASS-SPLICE), elastomeric bearings (BRASS-PAD), and luminaire poles (BRASS-POLE), as well as a module for the determination of wheel distribution factors (BRASS-DIST). The Office of Structures currently supports the use of BRASS-GIRDER. BRASS-GIRDER accomplishes the design and load capacity determination of highway bridges. The program utilizes finite element theory and AASHTO specifications, and accommodates straight steel, concrete, and timber beams. The system computes moments, shears, axial forces, deflections and rotations caused by dead loads, live loads, settlements and temperature changes. These actions are used to design or rate user-specified sections of the deck, girder and integral columns.

• **CANDE-2007 (Culvert ANalysis and DEsign)** is a public-domain finite element design and analysis tool for all types and sizes of buried structures (culverts).
• **CONSPAN LA** is a Windows-based program for the analysis and design of single-span and multiple-span bridges constructed with simple-span, prestressed concrete girders and made continuous by reinforcing the cast-in-place top deck with mild steel in regions of negative moment.

The program possesses a predefined library of strand and section types which can be modified by the user. Standard LFD and LRFD trucks can be selected from the live load library or a configuration can be manually entered to perform an automatic moving load analysis of the structure. Input wizards are used to define beam layout and material properties, dead loads, select specific limit states for analysis and design, customize load and resistance specified factors, select limiting stresses for concrete, and specify percentage of debonded or draped strands. The program checks design status at critical points for release and final stresses as well as for ultimate loads. It automatically generates straight or draped strand patterns for a specific beam. Cracking load criteria is also checked. Vertical and horizontal shear steel is designed as well as negative reinforcement in the deck and restraint moment connections at the piers.

• **Consplice**

Consplice is a Windows-based program for the analysis and design of spliced prestressed/precast bridge girders. Splices are cast-in-place with longitudinal post-tensioning. Available precast beams include l-girder, box beam, open box/bathtub beam, tee, or double-tee beam. The user can specify variable depth precast beam segments and end blocks at either or both beam ends. The tendon profile can be linear, general, or parabolic (two-, three- or four-span). Jacking can be specified from either or both ends and can be done in single or multiple stages.

The program can easily switch between English and metric unit systems. To save input time, there are built-in libraries for precast beam sections, prestressing stands, post-tensioning tendons and live load vehicles.

The program performs a time dependent analysis using either 1990 CEB-FIP, AASHTO LRFD or ACI-209 committee model codes for concrete creep, shrinkage, and steel relaxation effects. This analysis is dependent on the construction stage sequence. For each stage, the user specifies the duration, which elements are active (beam, slab, cast-in-place splices, post-tensioning, or support elements), and which loads are being applied. The program provides a graphical depiction of each stage.

The program automatically performs a moving live load analysis using AASHTO vehicles. The program can also analyze the structure for a temperature gradient (positive and negative).

A design check is done using either AASHTO LFD or LRFD specifications for ultimate moment, shear and service load stresses. The user can view the results of the analysis (moments, shears, stresses and deflections) in either tabular or graphical form. A capacity/demand ratio is calculated for ultimate moment and cracking load. The program can design or analyze vertical shear reinforcement. It can also design prestressing strand and debonding for variable support conditions at release.
DESCUS I (DESign and Analysis of CUrved I-Girder Bridge Systems) is an analysis and design (partial design) software for horizontally curved composite or noncomposite I-girder steel bridges. The user can specify the use of either WSD, LFD or LRFD (loading only) methods. The input can be in English or SI units. The bridge can be continuous and skewed over supports. The girders can have a high degree of curvature, can be nonconcentric, and may contain hinges.

The program models the bridge as a two-dimensional grid structure with three degrees of freedom at each nodal point. All dead load computations are performed automatically within the program to satisfy the construction conditions specified by AASHTO. The user can input additional dead loads as desired. All live load computations are also performed automatically where the AASHTO truck and lane loadings are applied to an influence surface previously generated for the entire bridge. Dynamic impact effects are also included. Arbitrary truck configurations can also be specified and analyzed.

The program output contains the positive and negative maximum moments, shear and torsion along with the corresponding primary and warping stresses for each girder and beam or truss diaphragm element. These maximums are given along with all AASHTO loading combinations. The output also includes deformations along each girder for dead load and maximum dead load plus impact along with the allowable recommended by AASHTO. The program will also perform rating calculations using either working stress rating (WSR), Load Factor Rating (LFR), or Load and Resistance Factor Rating (LRFR) methods.

DESCUS II possesses the same features and functions as DESCUS I, but was specifically written to analyze a horizontally curved structure composed of steel box sections.

ETCulvert will design and analyze a one-cell precast reinforced concrete culvert with prismatic members with or without a bottom slab in accordance with the design criteria in NYSDOT LRFD Bridge Design Specifications or AASHTO Standard Specifications for Highway Bridges.

Mathcad Professional is a general purpose computational tool. Mathcad allows text and math to be combined in the same document. Since the program uses real math notation, worksheets created in Mathcad look just like computations made with paper and pencil. Formulas in a Mathcad worksheet are “live” in the sense that if a change to a variable is made, all equations are recomputed automatically. Therefore, Mathcad worksheets can replace hand-calculations where changes are frequently necessary.

Mathcad has extensive computational ability. Equations can be solved numerically or symbolically. Two- and three-dimensional plots can be readily created. Mathcad can also handle variables and equations that have units associated with them. More advanced features include matrices and vectors, derivation and integration, built-in and user defined functions, solving blocks of equations, programming, and symbolic evaluation.
• **MDX** is an analysis, design, and rating software for horizontally curved and straight composite or non-composite steel I girders, box girders, or rolled shapes. The user can specify ASD, LFD, or LRFD design including use of standard live loads, user defined live loads (e.g. NYSDOT permit loads) or rail loadings. The software offers flexible nodal coordinate input feature to accommodate complex girder system framing plans, roadway profile, variable horizontal curvature, and skewed supports. The user can specify a line girder approach, grid analysis, or plate and eccentric beam finite analysis model for girder system analysis, design and load rating. The software can accommodate up to 20 spans and 60 girders of complex girder web profile of uniform and hybrid steel girders.

The MDX design and analysis software is also capable of performing design and analysis of various girder attachments such as bearing stiffeners, intermediate transverse web stiffeners, longitudinal web stiffeners, box girder bottom flange stiffeners, shear connectors, welds, bolted splices, and variety of bracing types and members. The software allows for slab pour sequence analysis with an output for camber data, stress, load deflections, and performance ratios to ensure compliance with AASHTO code.

The MDX design and analysis software complements other curved girder design and analysis software allowing designer higher level of confidence and comfort while reducing possible errors in design and analysis of horizontally curved girders.

• **MERLIN-DASH** *(Design, Analysis and Rating of Straight Girder Bridge Systems)* is a steel beam and plate girder design and analysis program that offers a Windows-based pull-down menu system, indexed output tables, the ability to perform a complete code check, rating capabilities, and graphics plots to interpret the numeric output. Various code specification methods are available in MERLIN-DASH including the AASHTO WSD, LFD, and LRFD alternates for design, analysis and rating. The user has the option of choosing either English or metric input and output. The program incorporates a wide range of live load analysis capabilities including standard and nonstandard AASHTO truck and lane loadings, interstate (or military) vehicle, and user-defined truck up to 20 axles where direction of travel may be specified. All dead load conditions, including dead load stage analysis, are given automatically for both composite and non-composite construction.

MERLIN-DASH can perform detailed steel designs for a wide range of configurations. Among the various features available to the user is design recycling, placement of lateral bracing, the shear/moment interactions, stiffener requirements, and minimum weight or minimum cost optimization. MERLIN-DASH also performs a detailed code check including a comparison of all actual stresses or stress resultants (moments, shears, etc.) and stress ranges to allowables generated automatically by the program. Supplementing all code check results, the program output lists the applicable code equation numbers, the code provisions, and the constants which are used to calculate the allowables. The results are given for all fatigue and nonfatigue details. Flags highlight all overstress conditions.

• **OPIS**

Opis is the analysis and design component of the Virtis/Opis software. The program uses AASHTO LRFD or LFD specifications for analysis and design. The program has a database component for storing all the input information (geometry, material properties, loads, etc.). This information is used by the different modules to analyze a structure. At the present time, OPIS has a module for steel girders and prestressed girders (Brass Girder).
• **RC-Pier LA**

RC-Pier LA is a Windows-based program for the analysis and design of reinforced concrete piers based on AASHTO LFD and LRFD codes. Wall, multi-column and hammerhead piers are all handled by the program. Footings can be either isolated, combined or strap and they can be either spread or on piles. The program can easily switch between English and metric unit systems.

The user specifies the geometry of the pier. Cap beams can be straight or tapered. Up to two lines of bearings can be specified. Columns can be rectangular or circular and can be tapered in either direction if rectangular. The program provides a three-dimensional visualization of the substructure.

Substructure dead loads are automatically calculated and the program can also generate live loads, wind loads and earthquake loads. Users can input bearing, column and cap loads for any load type. The user can also specify which load groups to include in the analysis. The results of the analysis can be viewed in tabular form or graphically for a specific load type or load group. These results include axial forces, shears, moments, displacements and rotations.

Reinforcement can be input by the user or automatically designed by the program. The cap is checked for flexure, shear, torsion, cracking and fatigue. Columns are checked for flexure and axial loads. Slender columns can be analyzed using P-delta or moment magnification methods. Interaction diagrams for the column can also be viewed. Footings are checked for flexure, one-way shear, two-way shear, crack control and fatigue. There is an optional strut-and-tie method for the analysis of hammerhead piers.

• **SEISAB (SEISmic Analysis of Bridges)** can be used to analyze simply-supported or continuous deck girder-type bridges for seismic response with no practical limitation on the number of spans or the number of columns at a bent. SEISAB contains both the single mode and multimode response spectrum analysis techniques included in AASHTO. In addition, earthquake restrainer units may be placed between adjacent structural segments. Horizontal alignments composed of a combination of tangent and curved segments are accommodated. Connections between the superstructure and the substructure and between adjacent superstructure segments at span hinges can be specified with either a keyword force release or by using bearing elements on a point by point basis. The flexibility of the soil and foundations at the abutments and column bottoms is included using stiffness coefficients or individual piles grouped into pile footings.

SEISAB has generating capabilities that will automatically build a model consistent with the modeling techniques used to conduct dynamic analyses. Seismic loadings in the form of acceleration response spectra are stored within SEISAB and may be referenced by the user, or a site specific spectrum can be utilized. A dead load analysis option can be requested for model verification or to obtain dead load forces for the Group VII loading. The user interacts with SEISAB by using the built-in menu system or by supplying an existing input file and data can be in either English or SI units.
- **STAAD-PRO (STructural Analysis And Design)** is a powerful software for static, dynamic, p-delta, nonlinear, buckling or cable analysis of structures. The program accepts truss, plane, floor, and space structural types. STAAD is capable of steel, concrete and timber design. The program uses a common language-based input format which can be created through an editor, a graphics input generator, or through CADD-based input generators. Modeling of the structure consists of two steps: identification of joints and nodes, and modeling of members or elements through specification of connectivity between joints. The structure is defined as an assemblage of elements. The graphics input generator facilitates viewing of structural models for both 2D and 3D situations, and allows the user to specify section properties, material constants, supports, loads, analysis/design requirements, and printing/plotting requirements. The program also allows member properties to be described using prismatic property specifications, standard steel shapes from the built-in section library, and through user-created steel tables, tapered sections, or assigned values. Graphical post-processing is available for verification of the model and display of the results, including display and plotting of structure geometry, deflected/mode shapes, bending moment/shear force diagrams, and stress contours. In addition to STAAD-Pro, the Office of Structures has also acquired the following ancillary programs:

  - **STAAD.beava**: Bridge Engineering Automated Vehicle Application, is used to automatically generate Live Load effects on 3-D models, using influence surfaces.

  - **STAAD.foundation**: a program for analysis and design of various types of foundations, such as isolated or combined spread footings, mat foundations, and pile footings.

  - **STAAD.etc**: a set of modules for analysis and design of structural components such as base plates, bolt groups, cantilever retaining walls, moment connections, masonry walls, rectangular footing, etc.

  - **Section Wizard**: creates custom shapes and calculates section properties. Can also calculate stress at any point of a cross section based on an applied Axial forces and Moments about principal axes.

- **VIRTIS**

  Virtis is the load rating component of the Virtis/Opis software. Virtis can provide bridge ratings using either AASHTO ASD or LFD specifications.