

First Edition, 2012

ISBN 978-81-323-3422-4

© All rights reserved.

Published by:

Research World

4735/22 Prakashdeep Bldg,

Ansari Road, Darya Ganj,

Delhi - 110002

Email: info@wtbooks.com

Table of Contents

Chapter 1 - Extreme Loading for Structures and NX (Unigraphics)

Chapter 2 - Advance Design and Advance Concrete

Chapter 3 - IOSO and Advance Steel

Chapter 4 - CANape, CANalyzer and AutoTURN

Chapter 5 - Optical Software

Chapter 6 - Finite Element Software

Chapter 7 - Geotechnical Engineering Software

Chapter 8 - Hydrology Models

Chapter-1

Extreme Loading for Structures and NX (Unigraphics)

Extreme Loading for Structures

Extreme Loading for Structures

Developer(s)	Applied Science International
Stable release	ELS Version 3.0 / September 3, 2009
Operating system	Windows
Type	Simulation software

Extreme Loading for Structures (ELS) - a commercial structural analysis software program based on the applied element method (AEM) for the automatic tracking and propagation of cracks, separation of elements, element collision, and collapse of structures under extreme loads. AEM combines features of Finite element method and Discrete element method simulation with its own solver capabilities for the generation of PC-based structural analysis.

History

2003

- Research and development related to the software begins with the formation of Applied Science International. The first release of ELS appears in the form of 2D analysis with structures modeled, loading scenarios applied, and results viewed.

2008

- Version 2.0 allows users to perform 3D analysis, though modeling is largely limited to 2D and restricted 3D functionality.

- The United States Department of Homeland Security assigns ELS Designation Status for Anti-terrorism under the SAFETY Act.

2009

- ELS version 3.0 is released with complete 3D functionality.

ELS Modeler

In the ELS modeler the user has the ability to model using a variety of default structural components such as columns, slabs, walls, windows, steel sections, reinforcement and 8-node objects. Pre-stressing, cables, pre-cracking and holes can also be added to models to reflect the required attributes of structures. To aid the user in modeling, attachments such as *.dxf, *.stl, *.dgn, and *.jpeg can be easily imported. Models can also be imported from several FEM based software programs such as Abaqus, ANSYS, Etabs, Gambit, Ideas, LS-DYNA, Nastran, Patran, SAP2000, and STAAD. ELS is Building Information Modeling (BIM) compatible with a plug-in for Autodesk Revit Structure, allowing users to import previously created structural components.

ELS Solver

The solver performs multi-stage static and dynamic analysis in 2D and 3D. Static loading types include concentrated loads, displacements, hydrostatic pressures, uniform pressures, moving loads, element removals, and line loads. Available dynamic loading types include concentrated loads, displacements, seismic loads, uniform pressures, element removals, and moving loads. ELS comes with a predefined material library with both linear and non-linear material models that include steel, concrete, reinforced concrete, glass, aluminum, elastic, and tension.

The **AEM solver** automatically calculates:

- Crack generation
- Plastic hinge formation
- Element separation
- Buckling, post buckling and large displacements
- Contact and collisions

ELS Output Viewer

Once the AEM solver begins, output can be immediately observed in the ELS viewer. A variety of animated diagrams, charts and files can be created for visual representation of the analysis. The ELS viewer can create the following exportable content:

- Animated diagrams:
 - Internal force
 - Envelope

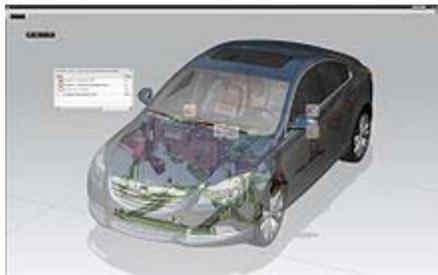
- Animated stress-strain contours
- Load displacement curves
- Animated charts available:
 - Load
 - Displacement
 - Stress
 - Strain
 - Time
 - Velocity
 - Acceleration
- Eigen modes
- Export to *.avi and *.bmp
- Through additional plug-ins ELS can export animation data to Autodesk 3ds Max and Maya for use in visual effects for television and film.

Applications

ELS is currently being utilized for structural engineering analysis functions such as structural vulnerability assessment & risk mitigation, progressive collapse analysis, blast analysis, demolition prediction analysis, impact analysis, seismic analysis, forensic engineering, performance based design, glass performance analysis product development and visual effects.

NX (Unigraphics)

NX (Software)



NX7 image of an automobile

NX 7

Developer(s)	Siemens PLM Software
Initial release	1973
Stable release	NX 7.5 / May 20, 2010; 10 months

	ago
Development status	active
Operating system	Windows XP or later (x86/x64), Unix-like, Mac OS
Available in	multi-language
Type	CAD/CAM/CAE/PLM
License	proprietary

NX, also known as **NX Unigraphics**, is an advanced CAD/CAM/CAE software package developed by Siemens PLM Software.

It is used, among other tasks, for:

- Design (parametric and direct solid/surface modelling)
- Engineering analysis (static, dynamic, electro-magnetic, thermal, using the Finite Element Method, and fluid using the finite volume method).
- Manufacturing finished design by using included machining modules.

NX is a direct competitor to Pro/ENGINEER, CATIA, SolidWorks and Autodesk Inventor.

History

1969: UNIAPT was released by a software company then called United Computing, UNIAPT was one of the world's first end-user CAM products.

1973: The company purchased the Automated Drafting and Machining (ADAM) software code from MGS in 1973. The code became a foundation for a product called UNI-GRAPHICS, later sold commercially as Unigraphics in 1975.

1976 McDonnell Douglas Aircraft buys Unigraphics.

1981: Unigraphics was released, marking the group's first true 3D modeling hardware and software offering.

1991: During a period of financial difficulties McDonnell Douglas sells Unigraphics to EDS which at that time is owned by General Motors. Unigraphics becomes GM's corporate CAD system.

1992: Over 21,000 seats of Unigraphics are being used worldwide.

1996: Unigraphics V11.0 is released. Added Industrial Design and Modeling enhancements including Bridge Surface, Curvature Analysis for Curve and Surfaces, Face Blends, Variable Offset Surface, etc. In the area of Assembly Modeling the new

capabilities included Component Filters, Faceted Representations, Clearance Analysis between multiple Components, etc. Also this release included a fully integrated Spreadsheet linked to Feature-Based Modeling.

2002 First release of the new "Next Generation" version of Unigraphics and I-deas, called NX. This will eventually bring the functionality and capabilities of both Unigraphics and I-DEAS together into a single consolidated product.

2007 Introduction of Synchronous Technology in NX 5.

A complete history exists on the Unigraphics Virtual Museum.

NX 7.5 is the current release.

Key Functions

- Design (Computer aided design)
 - Parametric Solid modeling (feature based and direct modeling)
 - Freeform surface modelling, Class A surfaces).
 - Reverse engineering
 - Styling and Computer-aided industrial design
 - Engineering drawing (Drafting)
 - Product and manufacturing information (PMI)
 - Reporting and Analytics, Verification and validation
 - Knowledge Reuse including Knowledge-based engineering
 - Sheet metal design
 - Assembly modelling and Digital mockup
 - Routing for Electrical wiring and Mechanical Piping
- Simulation (Computer-aided engineering)
 - Stress analysis / Finite element method (FEA)
 - Kinematics
 - Computational fluid dynamics (CFD) and Thermal analysis
- Manufacturing (Computer-aided manufacturing)
 - Numerical control (NC) programming

Supported operating systems and platforms

NX runs on Microsoft Windows, Mac OS, and Linux.

Architecture

NX uses Parasolid for its Geometric modeling kernel and D-Cubed as Associative engine for sketcher and assembly constraints as well as using JT (visualization format) for lightweight data and Multi-CAD.

Chapter-2

Advance Design and Advance Concrete

Advance Design

Advance Design is a computer-aided engineering (CAE) software application developed by GRAITEC and dedicated to structural analysis and design of reinforced concrete / steel structures and automated creation of design reports.

Features

Advance Design offers a complete environment for the static and dynamic analysis of 2D and 3D reinforced concrete and steel structures using the finite elements method.

Advance Design® is part of the BIM structural GRAITEC Advance® suite, automating the entire structural design process, from engineering design to detailing and fabrication.

Characteristics

- 3D Climatic generator: automatic generation of wind and snow loads.
- Static, dynamic, non-linear analyses, large displacements, generalized buckling, etc.
- Boundary conditions on planar elements.
- Multi-core finite elements engine.
- Results post-processing (graphical visualization of results, result curves, stresses diagrams, storing post-processing graphic views, reports with automatic update).
- Determination of 3D interaction curves on an arbitrary section.
- Advanced meshing engine: "grid" or Delauney mesh algorithm, finite elements with 3, 4, 6 or 9 nodes, progressive meshing, refinement tools, global mesh parameters.

Advance Design is available in two versions: one according to North American standards and one according to European standards.

Standard formats

- IFC
- CIS/2
- PSS
- SDNF
- DXF

Compatibility

- Windows 7, Windows Vista, Windows XP Pro (32-bit and 64-bit)

Specific functionalities

Customizable reports with automatic update. Advance Design includes an advanced report generator, which allows users to create standard or highly customized reports:

- Users can add different items to their reports (tables, texts, images, stored post-processing views, etc).
- Advance Design reports can be saved in different formats (DOC, RTF, TXT, PDF).
- During the modeling phase users may generate reports of the structure description (quantitative and qualitative data). After calculation, it is possible to generate reports containing result tables.
- The report generator takes into account defined by users. As a result, the reports content will refer only to the selected elements. Users are able to configure the look and structure of the document and also to input text and add images to the report.
- Users can define and save custom tables, in order to obtain the desired results in the calculation report.

Software interoperability

Advance Design integrates GRAITEC's technology, "GTC" (GRAITEC Transfer Center), a data synchronization technology that allows:

- Importing / exporting data to other Gritec software and standard formats: IFC 2.×3, CIS/2 (Referinta), SDNF.
- Multiple Advance Design users working simultaneously on the same project and synchronizing their models.
- Synchronization in Advance Design of modifications made by engineers in other Gritec software (e.g., section changes, addition of structural elements, etc.).

GTC is GRAITEC's solution for CAD / Design software interoperability and integration specialized in creating and handling a BIM (Building Information Model).

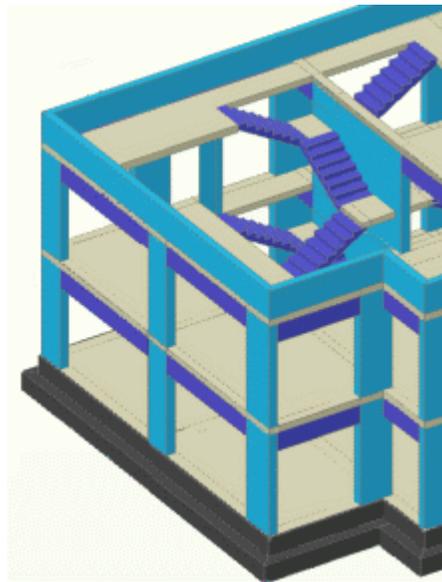
Release history

Official name	version	release	date of release
Advance Design 2011	2011	8	December, 2010
Advance Design 2010	2010	7	September, 2009
Advance Design 3.1.	3.1.	5	January, 2008
Advance Design 2.1.	2.1	4	January, 2007
Effel Advance 1.2 (previous name for Advance Design)	1.2	3	April, 2006
Effel Advance 1.1 (previous name for Advance Design)	1.1	2	June, 2005

Advance Concrete

Advance Concrete is a computer-aided design (CAD) software application developed by GRAITEC and used for modeling and detailing reinforced concrete structures in AutoCAD. Advance Concrete is used in the structural / civil engineering and drafting fields.

Features



Building with two floors, with stairs, modeled in Advance Concrete.

Advance Concrete is a software application integrated into AutoCAD, running as an AutoCAD plug-in. Advance Concrete functionalities are grouped in specific toolbars/ribbon tabs in the AutoCAD interface.

The application can be used for modeling and detailing different types of concrete structures, such as buildings, precast concrete elements, and also for civil engineering designs.

Advance Concrete uses AutoCAD technology: ObjectARX. This technology provides users with professional objects (beams, columns, slabs, bars, frames, stirrup bars) integrated into AutoCAD and on which most basic AutoCAD functions can be applied (stretch, shorten, copy, move, etc.).

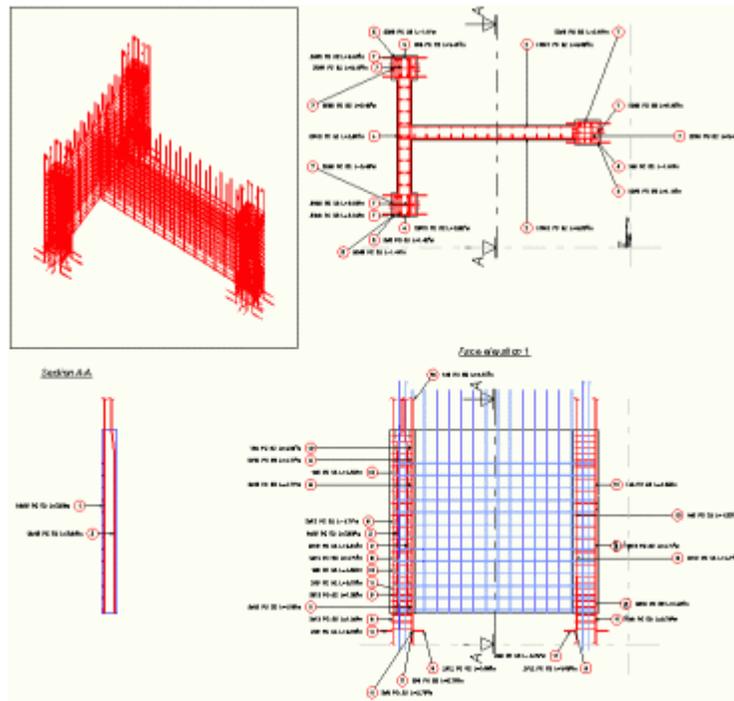
The Advance Concrete native file is the DWG file.

The main functions of Advance Concrete concern:

- 2D / 3D modeling of concrete structures;
- Automated drawings, with tools for automatic creation of sections, elevations, foundations, isometric views, etc.;
- Advanced reinforcement, with automatic creation and update functionalities and also with manual input tools;
- Creation of paper drawings;
- Automatic bill of materials.
- Multi-user modeling. Users can securely and simultaneously work on the same project through a shared database that stores the model data.

The program provides a working environment for creating 3D structural models from which drawings are created. The 3D model is created using Advance Concrete specific objects (structural elements, openings, rebars, etc.) and stored in a drawing (in DWG format). Once a model is complete, Advance Concrete creates all structural and reinforcement drawings using a large selection of tools for view creation, dimensions, interactive annotations, symbols, markings and automatic layout functions. Advance Concrete provides functionalities for automatic drawing updates based on model modifications.

Specific functionalities



Reinforcement drawing created in Advance Concrete.

One of the features of Advance Concrete is the “dynamic reinforcement” technology for the rapid reinforcement of concrete elements taking into account their context (type, parameters, connection with other elements). The user creates a so called dynamic reinforcement solution that integrates the reinforcement cage elements and properties: geometric information, local rules and standards for steel grade, reinforcement bar placement, concrete cover, etc. The reinforcement solutions can be used later for elements that have different sizes. The reinforcement elements adjust to the new dimensions and are taken into account at reinforcement drawing creation and lists. The user can save the reinforcement solution in an external file, which can be exchanged, downloaded, reused in any other projects etc.

Software compatibility

The application is compatible with the following operating systems and Autodesk platforms:

- Windows Vista, Windows XP Pro and Windows 7 (32 and 64 bits)
- AutoCAD 2007, 2008, 2009 32 and 64 bits, 2010 32 and 64 bits
- ADT 2007,
- AutoCAD Architecture 2008, 2009, 2010, 2011.

Software interoperability

Advance Concrete integrates GRAITEC's "GTC" (GRAITEC Transfer Center) technology, a data synchronization technology that allows:

- Importing / exporting data to other GRAITEC software and standard formats (IFC 2.x3)
- Several Advance Concrete users working simultaneously on the same project and synchronizing their models
- Synchronizing in Advance Concrete the modifications made by engineers in other GRAITEC software applications (e.g., section changes, addition of structural elements, etc.).

GTC is the GRAITEC solution for CAD/Design software interoperability and integration specialized in creating and handling a BIM (Building Information Model).

Release history

Official name version release date of release

Version 2011	2011	10	2010
Version 2010	2010	9	2009
Version 2009	2009	8	2009
Version 8.1	8.1	7	2008
Version 7.1	7.1	6	2006
Version 6.1	6.1	5	2005
Version 5.2	5.2	4	2004
Version 5.1	5.1	3	2003
Version 4.2	4.2	2	2002
Version 4.1	4.1	1	2001

Chapter-3

IOSO and Advance Steel

IOSO

IOSO (Indirect Optimization on the basis of Self-Organization) is a multiobjective, multidimensional nonlinear optimization technology.

IOSO approach

IOSO Technology is based on the response surface methodology approach. At each IOSO iteration the internally constructed response surface model for the objective is being optimized within the current search region. This step is followed by a direct call to the actual mathematical model of the system for the candidate optimal point obtained from optimizing internal response surface model. During IOSO operation, the information about the system behavior is stored for the points in the neighborhood of the extremum, so that the response surface model becomes more accurate for this search area. The following steps are internally taken while proceeding from one IOSO iteration to another:

- the modification of the experiment plan;
- the adaptive adjustment of the current search area;
- the function type choice (global or middle-range) for the response surface model;
- the adjustment of the response surface model;
- the modification of both parameters and structure of the optimization algorithms; if necessary, the selection of the new promising points within the search area.

History

IOSO is based on the technology being developed for more than 20 years by Sigma Technology which grew out of IOSO Technology Center in 2001. Sigma Technology is headed by prof . Egorov I. N., CEO.

Products

IOSO is the name of the group of multidisciplinary design optimization software that runs on Microsoft Windows as well as on Unix/Linux OS and was developed by Sigma Technology . It is used to improve the performance of complex systems and technological processes and to develop new materials based on a search for their optimal parameters. IOSO is easily integrated with almost any computer aided engineering (CAE) tool.

IOSO group of software consists of:

- IOSO NM: Multi-objective optimization;
- IOSO PM: Parallel multi-objective optimization;
- IOSO LM: Multilevel multi-objective optimization with adaptive change of the object model fidelity (low-, middle-, high fidelity models);
- IOSO RM: Robust design optimization and robust optimal control software;

Purpose

Performance improvement and design optimisation

IOSO NM is used to maximise or minimise system or object characteristics which can include the performance or cost of or loads on the object in question. The search for optimal values for object or system characteristics is carried out by means of optimal change to design, geometrical or other parameters of the object.

Search for optimal system management laws

It is often necessary to select or co-ordinate management parameters for the system while it is in operation in order to achieve a certain effect during the operation of the system or to reduce the impact of some factors on the system.

Identification of mathematical models

When the design process involves the use of any mathematical models of real-life objects, whether commercial or corporate, there is the problem of co-ordinating the experiment findings and model computation results. All models imply a set of unknown factors or constants. Searching for the optimal values thereof makes it possible to co-ordinate the experiment findings and model computation results.

Robust design optimization and robust optimal control

Introduction

Practical application of the numerical optimization results is difficult because any complex technical system is a stochastic system and the characteristics of this system have probabilistic nature. We would like to emphasize that, speaking about the stochastic properties of a technical system within the frame of optimization tasks, we imply that the important parameters of any system are stochastically spread. Normally it occurs during the production stage despite of the up-to-date level of modern technology. Random deviations of the system parameters lead to a random change in system efficiency.

An efficiency extreme value, obtained during the optimization problem while solving in traditional (deterministic) approach, is simply a maximum attainable value and can be considered as just conventional optimum from the point of view of its practical realization. Thus, one can consider two different types of optimization criteria. One of them is an ideal efficiency which can be achieved under the conditions of absolutely precise practical replication of the system parameters under consideration. Other optimization criteria are of probabilistic nature. For example: mathematical expectation of the efficiency; the total probability of assuring preset constraints; variance of the efficiency and so on. It is evident that the extreme of the one of these criteria doesn't guarantee the assurance of the high level of another one. Even more, these criteria may contradict to each other. Thus, in this case we have a multiobjective optimization problem.

IOSO robust design optimization concept

IOSO concept of robust design optimization and robust optimal control allows to determine the optimal practical solution that could be implemented with the high probability for the given technology level of the production plants. Many modern probabilistic approaches either employ the estimation of probabilistic efficiency criteria only at the stage of the analysis of obtaining deterministic solution, or use significantly simplified assessments of probabilistic criteria during optimization process. The distinctive feature of our approach is that during robust design optimization we solve the optimization problem involving direct stochastic formulation, where the estimation of probabilistic criteria is accomplished at each iteration. This procedure reliably produces fully robust optimal solution. High efficiency of the robust design optimization is provided by the capabilities of IOSO algorithms to solve stochastic optimization problems with large level of noise.

Advance Steel

Advance Steel is a CAD software application for 3D modeling and detailing of steel structures in AutoCAD and automatic creation of fabrication drawings, bill of materials and NC files, developed by GRAITEC.

Features

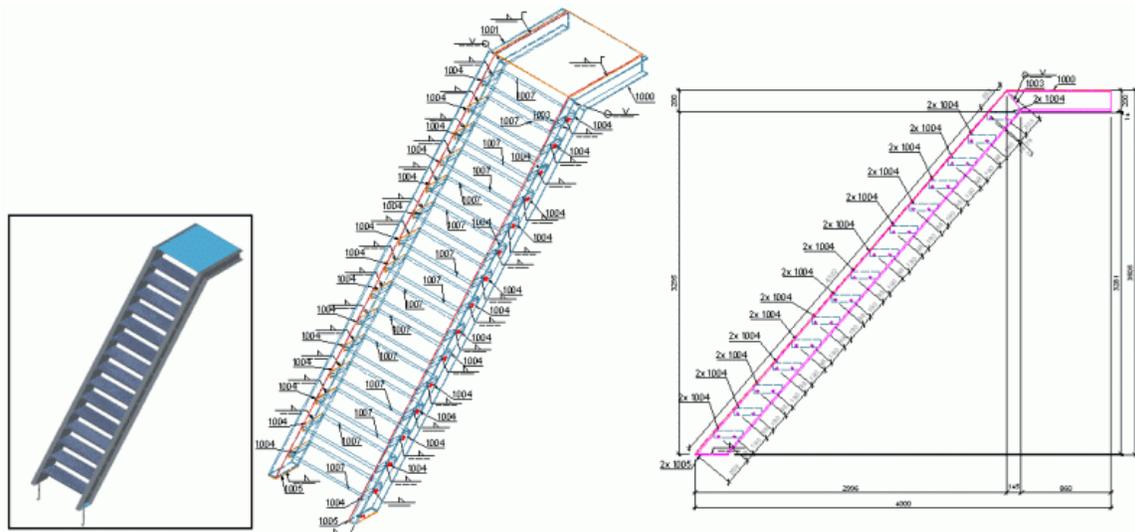
Advance Steel is integrated into AutoCAD which serves both as a graphic engine and an object oriented database for Advance Steel. The application supports all basic AutoCAD concepts and functions (snap points, grip points, copy, etc.).

The Advance Steel information is stored in DWG format.

The application includes AutoLisp (enhancing standard AutoLisp to include Advance Steel commands) and COM (VBA, C++) programming interfaces. This means that users can create their own customized macros for specialist requirements.

Advance Steel imports and exports to the following file formats:

- GTC;
- DWG;
- IFC. 2x3;
- CIS/2;
- SDNF;
- PSS;
- KISS (“Keep it Simple, Steel”)
- DSTV;
- DXF



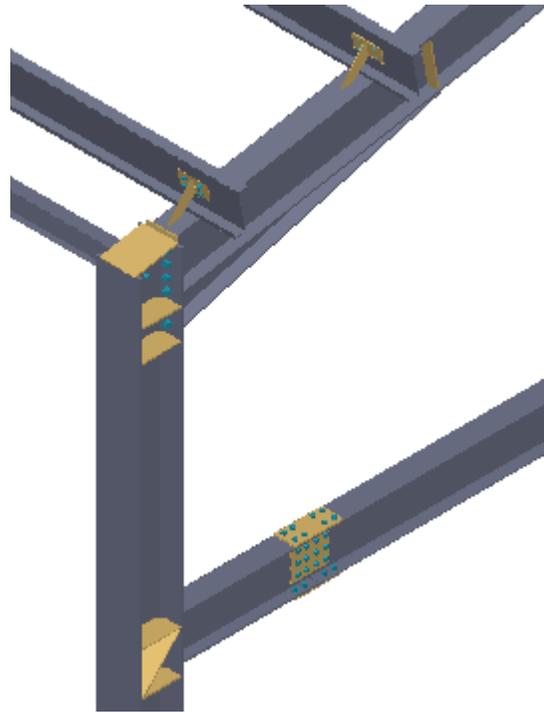
Detail views of a stair created using Advance Steel.

The main functions of Advance Steel concern:

- Creation of 3D model using a library of construction elements (i.e., beams, plates, bolts, welds, etc.).
- Sheet metal and plate work
- Advanced tools for element collision detection.
- Clear workshop drawings, automatically labeled and dimensioned.
- Checking the model in order to insure a correctly built 3D model and accurate bills of materials.
- Automatic creation of general arrangement and shop drawings, fabrication drawings, fitting drawings, isometric views and fabrication drawings.
- Drawing creation workflow management (revision control, automatic update, etc.)
- Automatic creation of lists / bills of materials and NC files.
- Multi-User Technology - all users involved in a project can work simultaneously and securely on the same model, without errors.

Advance Steel provides instruments for modeling complex structures such as straight and spiral stairs, railings, ladders, etc. The program creates all necessary documents (including NC files) for the stair fabrication.

Specific features



Structure with connections, modeled in Advance Steel.

Parametric steel connections

Advance Steel has a library of more than 300 preset parametric steel connections to connect Advance elements grouped in the following categories: beam end to end joints, base plate joints, general bracing joints, cantilever beam to column joints, plate joints, clip angle joints, pylon joints, tube brace joints, purlin joints, stiffener joints, turnbuckle bracings, etc.

The user creates all connecting elements by a single operation. At the same time, the connected elements are processed (shortened, coped, etc.). The software allows users to customize the connections:

- Set the parameters of the joint;
- Process the connected elements;
- Transfer the properties from one steel connection to another;
- Update the steel connection;

Joint Design engine

The software dimensions and checks joints according to Eurocodes 3 standards and AISC North American standards. A design report can be created.

Drawing styles

Based on a 3D-model, dimensioned and labeled 2D general arrangement and shop drawings can be automatically created using drawing styles. The drawings are created in separate DWG files; however they are linked to track changes. Thus, the drawings can be updated after any model modifications and the drawing revision can be managed.

The software has a variety of predefined drawing styles for the creation of general arrangement drawings and shop drawings for single parts and assemblies. A drawing style is a set of rules used to create a detail drawing and defines the elements that are displayed including labeling and dimensioning preferences.

Drawing styles provide the option to automatically create drawings and to modify the layout exactly to user requirements. Drawing styles are used in a similar way to AutoCAD dimension styles, line styles, etc. The predefined drawing styles are different for each installation and country. Custom drawing styles can also be defined.

Software compatibility

The application is compatible with:

- Windows 7, Windows Vista, Windows XP Pro (32-bit and 64-bit);
- AutoCAD 2007, 2008, 2009, 2010, 2011;
- ADT 2007;
- AutoCAD Architecture 2008, 2009, 2010, 2011.

Software interoperability

Advance Steel integrates GRAITEC's data synchronization technology, "GTC" (GRAITEC Transfer Center). This technology offers:

- Import/Export data in standard formats: CIS/2, SDNF, PSS, IFC 2.x3.
- Multiple Advance Steel users that work simultaneously on the same project and synchronize their models.
- Synchronization in Advance Steel of modifications made by engineers in other GRAITEC software (section changes, addition of structural elements, etc.).

GTC is Graitec's solution for CAD / Design software interoperability and integration specialized in creating and handling a BIM (Building Information Model).

Release history

Official name version release date of release

Version 2011 2011 8 December, 2010

Version 2010 2010 7 September, 2009

Version 2009 2009 6 November, 2008

Version 8.1	8.1	5	December, 2007
Version 7.1	7.1	4	March, 2007
Version 6.1	6.1	3	December, 2005
Version 5.1	5.1	2	July, 2004
Version 4.2	4.2	1	July, 2003

Chapter-4

CANape, CANalyzer and AutoTURN

CANape

CANape



Developer(s)	Vector Informatik
Initial release	1996
Stable release	9.0 / July 22, 2010; 8 months ago
Operating system	Windows XP, Vista, 7
Type	Data Acquisition
License	Proprietary

CANape is a software tool from Vector Informatik. This development software, widely used by OEMs and ECU suppliers of automotive industries, is used to calibrate algorithms in ECUs at runtime.

Description

In calibrating ECUs, control behavior is modified for different vehicle models or variants. This is not done by making changes to the program code, but instead by changing the ECU's parameterization. This involves the use of measurement and calibration systems such as CANape in the laboratory, on test benches or in vehicles during test trials. To evaluate the effects of changes to parameters, development engineers access the relevant process variables with conventional measurement technology at sensors and actuators and read out the data from the ECU. Measured data internal to the ECU, e.g. intermediate results of a computational function, can be accessed via the ECU's standard interfaces (CAN, FlexRay, LIN, Ethernet) via proprietary hardware interfaces. A typical use case for calibration with CANape is online calibration. This involves modifying parameters directly in the ECU. The resulting control characteristic can be measured and checked directly. Using this approach, measured data from the ECU or physical measurement variables on or in the vehicle can be precisely analyzed to determine the effects of each individual change.

Features

All functions required to modify parameter values are implemented as standard features in CANape: Measuring, calibrating, administering, evaluating, flashing. CANape also enables symbolic access to data and functions accessible via the diagnostic protocol, and it supports calibration over XCP on FlexRay. Options extend the functional features of CANape by enabling access to models at runtime in Simulink, optical verification of object detection algorithms in developing driver assistance systems, visualization of a vehicle's GPS position and an ASAM MCD3 interface.

Versions

Version 1.0 was released in 1996. Up to Version 6.0 the product was known as CANape Graph. The latest version of CANape is 9.0 (revision level: July 2010).

Supported Standards

Internal ECU parameters are accessed via standardized measurement and calibration protocols such as CCP (CAN Calibration Protocol) and XCP (Universal Measurement and Calibration Protocol). CANape was the first measurement and calibration tool to enable access over XCP on CAN and XCP on FlexRay.

Supported ASAM standards, status as of April 2010::

- AE MCD-1(XCP) V1.0

- XCP on CAN Interface Reference V1.0
- XCP on Ethernet Interface Reference V1.0
- XCP on FlexRay Interface Reference V1.0
- XCP on SxI Interface Reference V1.0
- XCP on USB Interface Reference V1.0
- AE MCD-1(CCP) V2.1
- AE MCD-2MC(ASAP2/A2L) V1.5 und V1.6
- AE MCD-2D(ODX) V1.1.4, V1.1.5, V2.0.1, V2.1.0, V2.2.0
- AE MCD-2(FIBEX) V1.1, V1.2, V2.0.0, V2.0.1
- AE MCD-3 V1.0, V2.0
- COM/DCOM Interface Reference V1.0, V2.0
- ASAP3 (Interface Specification) V2.1

Other supported standards:

- KWP2000 on K-Line
- ISO 14230 (KWP2000 on CAN) and ISO 14229 (UDS)
- FlexRay Communications System Specification V 2.1
- CAN with DBC description format
- LIN 1.0, 1.3, 2.0, 2.1

If a development task requires a high measurement data throughput of up to 5MByte/s, Vector's VX1000 System can be used to access data over microcontroller-specific data trace and debug interfaces.

CANalyzer

	CANalyzer
Developer(s)	Vector Informatik
Initial release	1992
Stable release	7.5 / December 1, 2010; 3 months ago
Operating system	Windows XP, Vista, 7
License	Proprietary

CANalyzer is an analysis software tool from Vector Informatik GmbH. This development software is widely used, primarily by automotive and electronic control unit suppliers, to analyze the data traffic in serial bus systems. The most relevant bus systems here are CAN, LIN, FlexRay, Ethernet and MOST as well as CAN-based protocols such as J1939, CANopen, ARINC 825 and many more.

Description

Vector first offered CANalyzer on the market in 1992 – the world’s first CAN software tool. CANalyzer has been continually updated since then, and today it is considered the world’s leading analysis tool for CAN buses. Besides its primary field of application, which is in-vehicle electronic networking in the automotive industry, CANalyzer is also used in many other industries such as rail transportation, heavy-duty vehicles, special-purpose vehicles, avionics, medical technology and many more. In addition to its robust bus monitoring functionality, CANalyzer also contains many stimulation and analysis functions for triggering and analyzing message traffic and data contents. These functions are incorporated in a measurement setup. Users can configure and extend the tool’s functionality with an integrated compiling programming language. Data is displayed and evaluated in both raw and symbolic formats. Back in 1992, Vector had already developed the DBC data format, which has become the de-facto standard in the automotive industry for exchanging CAN descriptions. Relevant standards are supported for other bus systems as well, such as FIBEX for FlexRay, LDF for LIN and EDS/DCF/XDD for CANopen.

Versions

Version 1.0 was released in 1992. The current version of CANalyzer is 7.5 (Revised: December 2010). Program Levels CANalyzer is available in different variants. They relate to the software’s functional features (fundamental, expert and professional), supported bus systems (CAN, FlexRay, etc.) and supported higher-level protocol options (SAE J1939, CANopen, etc.).

AutoTURN

	AutoTURN
Developer(s)	Transoft Solutions Inc.
Initial release	1991
Stable release	AutoTURN 6.0
Operating system	Microsoft Windows
Platform	Bentley Systems MicroStation / Autodesk AutoCAD
Type	CAD
License	Proprietary software

AutoTURN is a third-party 2-dimensional CAD software released for the AutoCAD or MicroStation platforms, developed and sold by Transoft Solutions Inc. AutoTURN

software can only be used on computers that run Microsoft Windows operating system. Among a number of things, it analyses and simulates swept path vehicle maneuvers.

History

- AutoTURN was originally developed for Transoft Solutions Inc. in 1991. AutoTURN was designed as the practical application of a University of Calgary civil engineering graduate studies thesis by Milton Carrasco. The first version of AutoTURN was released for use on Autodesk's AutoCAD Release 9.
- In 1992, AutoTURN 2.0 was released and was the first released version of AutoTURN compatible with Bentley Systems MicroStation Unix. The libraries of vehicles available for swept path simulation and analysis was limited and vehicle simulations could only be carried out in a forward direction. AutoTURN 2.0 was compatible with AutoCAD Release 10 and then Unix version of MicroStation V 3.4.
- AutoTURN 3.0 was released in 1993, less than 1 year after AutoTURN 2.0. The 3.0 release version of AutoTURN included many upgrades over its predecessor. This version of AutoTURN included larger vehicle libraries, and a supplement for use with aircraft (This supplement would become expanded into a separate software called AutoTURN Aircraft in 1995). Among other improvements, AutoTURN 3.0 enabled users to run simultaneous simulations of two vehicles in forward and reverse directions in a stop motion (frame by frame) animation. Simulations could only be done from pre-drawn paths.
- In 2000 the release of AutoTURN 4.0 accounted for international users, it was the first version of AutoTURN available in multiple languages. Among other improvements, AutoTURN 4.0 included expanded international libraries of vehicles. To incorporate the many new capabilities it was necessary to make significant changes to the program's visual interface and functionality. The change in interface included a new toolbar. Generate simulation tools were introduced so users didn't first have to draw the vehicle path in order to generate a simulation. AutoTURN 4 introduced the concepts of Active Path and Active Simulation allowing for repetition of various functions without repeated re-selecting of objects. Like the previous release versions, AutoTURN 4.0 contained a supplement for aircraft.
- AutoTURN 5.0's release in 2004, among many other functions, introduced Heuristic algorithms for tracking, creation of simulations on spline elements, and improved layer management for simulations. The standard vehicle library file was changed to a searchable database of vehicles based on international or custom standards. Adding to those features AutoTURN 5.1 introduces the ability to set Steering Linkage ratios between the front and rear steering axle groups. Additionally, in the 5.1 version could continuously loop simulations.

- The 2008 release of AutoTURN 6.0 added additional simulation functions. It builds on version 5.1's abilities to loop simulations by adding the ability to export simulations to InVision (a separate software application) in order to create video clips. Plan view and chamfered vehicles can be used for simulations and or modified in this version.

Release history

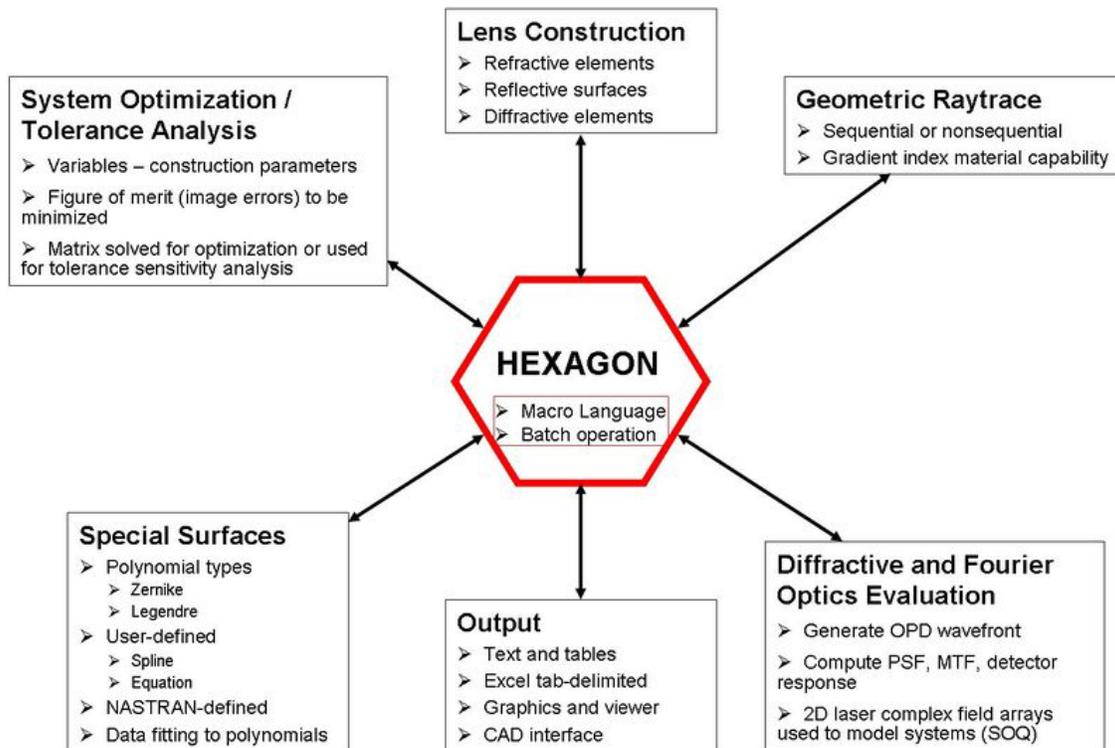
Name	Version	Date Released	Comments
AutoTURN 1.0	1.0	Nov, 1991	Initial release, available for Autodesk's AutoCAD Release 9
AutoTURN 2.0	2.0	July, 1992	First version to run on Bentley Systems' MicroStation (Unix)
AutoTURN 3.0	3.0	May, 1993	First version to include an aircraft supplement
AutoTURN 4.0	4.0	Oct, 2000	Graphical based simulations
AutoTURN 4.1	4.1	Aug, 2002	AutoCAD MDI compliant, 2002, MicroStation V8, Windows 95 98 NT 2000 XP compatible
AutoTURN 4.2	4.2	Jun, 2003	Chinese language support
AutoTURN 5.0	5.0	Nov, 2004	Introduced Heuristic algorithms for tracking
AutoTURN 5.1	5.1	Apr, 2006	Ability to set Steering Linkage ratios between the front and rear steering axle groups
AutoTURN 6.0	6.0	Apr, 2008	Library database search, customizable vehicles, InVision support
AutoTURN 6.1	6.1	Apr, 2009	SQL Express Database engine, AutoCAD 2010 and MicroStation V8i compatibility, Microsoft Window Vista - 64 bit compatibility

Chapter-5

Optical Software

HEXAGON

HEXAGON is a proprietary optical engineering software program used to design and analyze a wide variety of optical lens systems. There is an extensive interface capability with other optical design programs (CodeV, Zemax, and OSLO); with mechanical CAD programs (Pro/Engineer, AutoCAD, and I-DEAS); and the structural CAD program NASTRAN. Data can be easily exported to and imported from MATLAB and Excel. Graphical output can be vector graphics, viewed with a built-in viewer or output to WMF format for inclusion in Word or PowerPoint, or bit-map graphics (BMP) image files. The program name derives from: Handy EXpert Application for General-purpose Optical aNalysis. Input is entirely command driven, allowing unattended batch operation; a built-in Macro programming language allows the construction of a simple new 'command' that can execute other commands in a complex manner. Users store their own previously written macros in a macro libraries (a single disk file), and there is an extensive 'permanent' macro library available for all users. The diagram below shows some of the features of the HEXAGON software:



HEXAGON features roadmap

History

In the 1970's Hughes Aircraft Company acquired the software code ACCOS-V and extensively upgraded it while maintaining the command structure: entry consists of a command word (case insensitive) up to six letters long, followed by an optional qualifier word up to eight letters long, then either a text field or a numeric field containing up to five numbers. The new code was named HEXAGON (the beginning acronym meaning Hughes EXecutable) and many optical designers used interactive terminals to run the program on an IBM 4341 mainframe computer. In the 1980's the program expanded its capabilities in optimization, tolerance analysis, and physical optics; this later capability is a subset of HEXAGON named SOQ, for System Optical Quality, that models coherent laser systems using a two-dimensional complex field array. In the 1985 Hughes was sold to General Motors and renamed GMH; in the 1997 GMH was acquired by Raytheon Company. Currently Raytheon has exclusive use of the proprietary software code.

Optics Software for Layout and Optimization

Optics Software for Layout and Optimization (OSLO) is an optical design program originally developed at the University of Rochester in the 1970s. The first commercial version was produced in 1976 by Sinclair Optics. Since then, OSLO has been rewritten

several times as computer technology has advanced. In 1993, Sinclair Optics acquired the GENII program for optical design, and many of the features of GENII are now included in OSLO. Lambda Research Corporation (Littleton MA) purchased the program from Sinclair Optics in 2001 .

The OSLO software is used by scientists and engineers to design lenses, reflectors, optical instruments, laser collimators, and illumination systems. It is also used for simulation and analysis of optical systems using both geometrical and physical optics. In addition to optical design and analysis, OSLO provides a complete technical software development system including interactive graphics, math, and database libraries.

Applications

OSLO provides an integrated software environment that helps complete contemporary optical design. More than a lens design software, OSLO provides advanced tools for designing medical instrumentation, illuminations systems and telecommunications equipment, to name just a few typical applications. OSLO has been used in a multitude of optical designs including holographic systems , anastigmatic telescopes , gradient index optics , off-axis refractive/diffractive telescopes . the James Webb Space Telescope , aspheric lenses , interferometers , and time-varying designs .

Capabilities

OSLO is primarily used in the lens design process to determine the optimal sizes and shapes of the components in optical systems. OSLO has the capability of modeling a wide range of reflective, refractive and diffractive components. In addition, OSLO is used to simulate and analyze the performance of optical systems. OSLO's CCL (Compiled Command Language), which is a subset of the C programming language, can be used to develop specialized optical and lens design software tools for modeling, testing, and tolerancing optical systems.

OSLO has many unique features, for instance slider wheels. This feature allows users to affix up to 32 graphical sliders providing callbacks to default or user-supplied routines that perform evaluation or even full optimization iterations when a slider is moved. Some examples in the use of these slider wheels to design telescopes are provided by Howard.

Compatibility

OSLO works with other software products using a DDE (Dynamic Data Exchange) Client/Server interface. This enables the program to work with products such as MATLAB to create a multi-disciplinary environment, such an environment was used to design and analyze the Thirty Meter Telescope (TMT) .

Editions

OSLO is available in one educational and 3 commercial editions.

Free Educational Product

- OSLO EDU

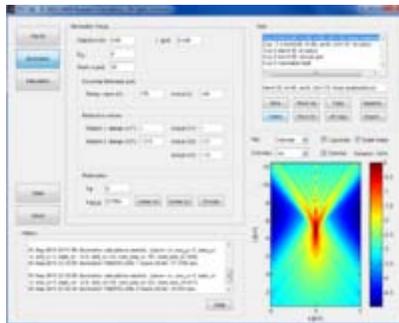
Tutorials, written specifically for beginners, on how to download and use this software are available from Ancient and Modern Optics and Steve Fejes

Commercial Products

- OSLO Light
- OSLO Standard
- OSLO Premium

PSF Lab

PSF Lab



PSF Lab on Windows 7 (x64)

Developer(s)	Michael J. Nasse, Jörg C. Woehl
Stable release	3.0 / September 24, 2010
Operating system	Microsoft Windows (x86, x64), Mac OS
Type	Optical software
License	Copyright (c) 2010, UWM Research Foundation (free for academic/non-commercial use)

PSF Lab is a software program that allows the calculation of the illumination Point Spread Function (PSF) of a confocal microscope under various imaging conditions. The calculation of the electric field vectors is based on a rigorous, vectorial model that takes polarization effects in the near-focus region and high numerical aperture microscope objectives into account.

The polarization of the input beam (assumed to be collimated and monochromatic) can be chosen freely (linear, circular, or elliptic). Furthermore, a constant or Gaussian shaped input beam intensity profile can be assumed. On its way from the objective to the focus, the illumination light passes through up to three stratified optical layers, which allows the simulation of an immersion oil/air (layer 1) objective that focusses light through a glass cover slip (layer 2) into the sample medium (layer 3). Each layer is characterized by its (constant) refractive index and thickness. PSF Lab can also simulate microscope objectives that are corrected for certain refractive indices and cover slip thicknesses (design parameters). Thus, any deviations from the ideal imaging conditions for which the objective was designed for are properly taken into account.

The following optical parameters can be selected:

- Input beam
 - Wavelength
 - Gaussian profile filling parameter (0 = constant profile)
 - Polarization (linear, circular, elliptic)
- Outputs
 - Individual field components
 - Squared field components
 - Intensity
- Microscope objective
 - Numerical aperture
- Optical media
 - Refractive index (design and actual)
 - Thickness (design and actual)
 - Depth (focus position within medium 3)

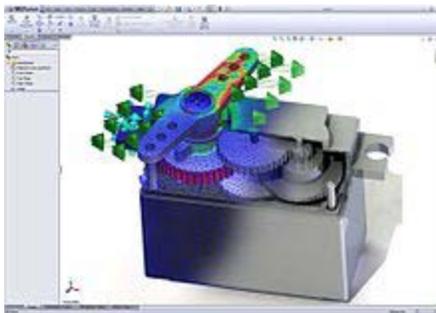
The program calculates only 2D section of the PSF, but several calculations can be stacked (with a third party program) to obtain the full 3D PSF. Calculations are organized in "sets", each with its own set of parameters. Loops can be set up such that PSF Lab calculates one or several sets, increasing the resolution of the calculated images in each new iteration. The resulting image is displayed in PSF Lab in linear or logarithmic color scale with user-selectable color map, and the intensity, individual field components, or squared field component distributions can be exported into various formats (data formats: .mat, .h5 (HDF5), .txt (ASCII); image formats: .fig, .ai, .bmp, .emf, .eps, .jpg, .pcx, .pdf, .png, .tif).

Chapter-6

Finite Element Software

NEi Fusion

NEi Fusion



Servo Motor with In-Flight Loading Modeled in NEi Fusion with NEi Nastran

-->

Developer(s)	NEi Software
Operating system	Windows XP, Windows Vista
Type	Computer Aided Engineering (CAE) software

NEi Fusion is a finite element analysis program sold by NEi Software that is used by engineers to build and analyze models of parts and assemblies of various products. NEi Fusion digital simulation software virtually applies forces, pressures, vibration, acceleration loads, or thermal conditions to 3D models of parts, structures, and assemblies and obtains results of various engineering parameters like deformation, stresses, strains, temperature distributions, and modal shapes the design will experience.

The results which range from tables of data to output like contour plots and animations provide engineering insight; for example, result visualizations like color-coded, contour plots help deepen understanding of physical phenomena in complex geometry. NEi Fusion consists of a 3D parametric CAD modeler powered by SolidWorks with NEi Nastran finite element analysis solvers. NEi Fusion runs on Microsoft Windows and provides CAD modeling, import and meshing tools.

Application

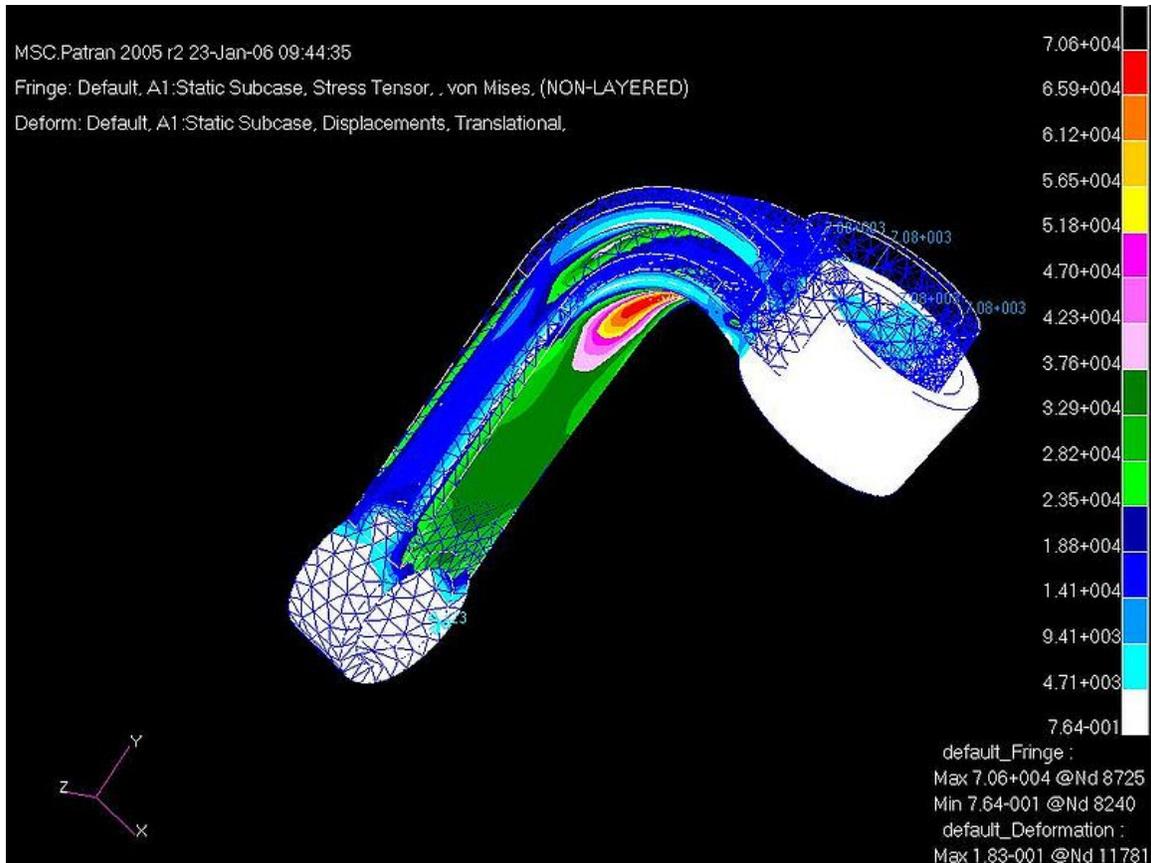
Finite element analysis software is typically used to improve the engineering design process by identifying potential problem areas, reducing development man-hours by eliminating portions of costly prototyping and testing, spurring innovation by allowing a way to evaluate different designs and materials, and providing a tool for optimizing designs early in the development cycle.

Present day

NEi Fusion 2.1, was released in February 2010. It now includes the following enhancements:

- Thermal Stress
- Nonlinear Transient Response
- Variable Loading
- XY Plot Enhancements
- Beam/Bar Enhancements
- Automated Bolted Joints
- Failure Theories
- Tree Simplification Enhancement
- Multi-Surface Contact Surface Selection

Nastran



Example of MSC.Nastran analysis result

NASTRAN is a finite element analysis (FEA) program that was originally developed for NASA in the late 1960s under United States government funding for the Aerospace industry. The MacNeal-Schwendler Corporation (MSC) was one of the principal and original developers of the public domain NASTRAN code. NASTRAN source code is integrated in a number of different software packages, which are distributed by a range of companies.

History

The 1964 annual review of NASA's structural dynamics research program revealed that the research centers were separately developing structural analysis software that was specific to their own needs. The review recommended that a single generic software program should be used instead. In response, an ad hoc committee was formed. The committee determined that no existing software could meet their requirements. They

suggested establishing a cooperative project to develop this software and created a specification that outlined the capabilities for the software.

A contract was awarded to Computer Sciences Corporation (CSC) to develop the software. The name of the program is an acronym formed from NASA Structural Analysis. The NASTRAN system was released to NASA in 1968. In the late 1960s, the MacNeal-Schwendler Corporation (MSC) started to market and support its own version of NASTRAN, called MSC/NASTRAN (which eventually became MSC.Nastran). The original software architecture was developed by Joe Mule (NASA), Gerald Sandler (NASA), and Stephen J. Burns (University of Rochester).

NASTRAN software application was written to help design more efficient space vehicles such as the Space Shuttle. NASTRAN was released to the public in 1971 by NASA's Office of Technology Utilization. The commercial use of NASTRAN has helped to analyze the behavior of elastic structures of any size, shape, or purpose. For example, the automotive industry uses the program to design front suspension systems and steering linkages. It is also used in designing railroad tracks and cars, bridges, power plants, skyscrapers, and aircraft. The program alone was estimated to have returned \$701 million in cost savings from 1971 to 1984. NASTRAN was inducted into the U.S. Space Foundation's Space Technology Hall of Fame in 1988, one of the first technologies to receive this prestigious honor.

The NASTRAN program has evolved over many versions. Each new version contains enhancements in analysis capability and numerical performance. In addition, many errors from previous versions are corrected. In one notorious case, an internal error in NASTRAN was identified as responsible for the 1991 collapse of the Sleipner A offshore platform. Today, NASTRAN is widely used throughout the world in the aerospace, automotive and maritime industries. It has been claimed that NASTRAN is the industry standard for basic types of analysis for aerospace structures, e.g. linear elastic static and dynamic analyses.

In November 2002 MSC Software reached a final agreement with the FTC to resolve an antitrust case against the company in connection with two acquisitions of rival CAE vendors, Universal Analytics, Inc. (UAI) and Computerized Structural Analysis & Research Corp. (CSAR). The FTC had alleged the acquisitions represented anticompetitive activities. Under the terms of the settlement, MSC divested a clone copy of its current Nastran software. The divestiture was through royalty-free, perpetual, non-exclusive licenses to UGS Corporation. UGS Corporation was acquired by Siemens in 2007.

Commercial versions of NASTRAN are currently available from MSC Software, NEi Software (NEi Nastran) and Siemens PLM Software (NX Nastran). Siemens AG purchased the former UGS Corporation from private equity concerns and their rights to the commercial version of NX NASTRAN in 2006.

Software architecture

NASTRAN is written primarily in FORTRAN and contains over one million lines of code. NASTRAN is compatible with a large variety of computers and operating systems ranging from small workstations to the largest supercomputers.

NASTRAN was designed from the beginning to consist of several modules. A module is a collection of FORTRAN subroutines designed to perform a specific task—processing model geometry, assembling matrices, applying constraints, solving matrix problems, calculating output quantities, conversing with the database, printing the solution, and so on. The modules are controlled by an internal language called the Direct Matrix Abstraction Program (DMAP).

Each type of analysis available is called a solution sequence.

Some of the most common solution sequence codes are:

- 101 - Linear Static
- 103 - Modal
- 105 - Buckling
- 106 - Non-Linear Static
- 107 - Direct Complex Eigenvalue
- 108 - Direct Frequency Response
- 109 - Direct Transient Response
- 110 - Modal Complex Eigenvalue
- 111 - Modal Frequency Response
- 112 - Modal Transient Response
- 129 - Nonlinear Transient
- 144 - Static Aeroelastic Analysis
- 145 - Flutter / Aeroservoelastic analysis
- 146 - Dynamic Aeroelastic Analysis
- 153 - Non-Linear static coupled with heat transfer
- 159 - Nonlinear Transient coupled with Heat transfer
- 187 - DDAM
- 200 - Design Optimization and Sensitivity analysis
- 400 - Non-Linear Static and Dynamic (implicit) (MSC.NASTRAN native, supersedes 106, 129, 153 and 159 - part of MD.NASTRAN)
- 600 - Non-Linear Static and Dynamic (implicit) (front end to MSC.Marc - part of MD.NASTRAN)
- 601 - Implicit Non-Linear (Adina for NX Nastran)
- 700 - Explicit Non-Linear (LS Dyna plus MSC.Dytran - part of MD.NASTRAN)
- 701 - Explicit Non-Linear (Adina NX Nastran)

Associated Software

NASTRAN is primarily a solver for finite element analysis. It does not have functionality that allows for graphically building a model or meshing. All input and output to the program is in the form of text files. However, multiple software vendors market pre- and post-processors designed to simplify building a finite element model and analyzing the results. These software tools include functionality to import and simplify CAD geometry, mesh with finite elements, and apply loads and restraints. The tools allow the user to submit an analysis to NASTRAN, and import the results and show them graphically. In addition to pre- and post-processing capabilities, several Nastran vendors have integrated more advanced nonlinear capabilities into their Nastran products.

NASTRAN Options

The following software options, based on NASTRAN original source code are available:

- MSC.Nastran and MD.Nastran
- NASTRAN-xMG (acquired by MSC Software)
- NEi Nastran (A PC/Linux-based version of the original NASTRAN source code)
- NX NASTRAN (acquired by Siemens PLM)
- Nastran distributed by the Open Channel Foundation

MSC.Nastran

MSC Nastran is the abbreviation of MacNeal-Schwendler Corp set up by two developers that worked on a NASA contracted to create a general purpose FEA for portability of government programs. Hence, MSC is a pioneer in simulation solutions to improve time-to-market. The company's solutions allow manufactures to accurately predict how their designs will behave in their intended environments, without having to build and test multiple physical prototypes. MSC Software has deep roots in the aerospace, automotive and heavy machine industries, where leading companies rely on MSC Software to build their products.

In addition to application specific tools, MSC Software also offers integrated enterprise solutions (SimXpert, SimDesigner, and SimManager) to enable the extended enterprise to leverage the advantages of enterprise-level simulation.

NASTRAN-xMG

Built from the same base code as the original NASTRAN software created by NASA that includes the original NASTRAN architecture and DMAP language, NASTRAN-xMG provides unlimited problem size, high-speed solver technology, and substructuring analysis options. The principal founder of the company is Dr. Richard H. MacNeal.

NEi Nastran

NEi Nastran is a general purpose finite element analysis solver used to analyze linear and nonlinear stress, dynamics, and heat transfer characteristics of structures and mechanical components. It is available on a variety of platforms including 32/64-bit Windows and Linux.

NX Nastran

NX Nastran resulted from an FTC action against MSC Corporation for alleged antitrust activity. The antitrust settlement allowed the 2003 purchase of a royalty-free, perpetual license for the MSC.Nastran 2001 source code by UGS. UGS was previously the Unigraphics division of EDS. Nastran's CAE capabilities were added to NX Unigraphics' CAD and other components to form the *EDS/UGS PLM Solutions* product life-cycle management suite. This product line was acquired by SIEMENS in 2007 and became *SIEMENS PLM Software*.

The purchase included a perpetual, worldwide, free of royalties, nonexclusive license of the software program MSC.Nastran v2001, other assets related to the software, and all copyright and trademark rights acquired as a result of MSC's acquisitions of Universal Analytics Inc. and Computerized Structural Analysis & Research Corp. in 1999.

OCF Nastran

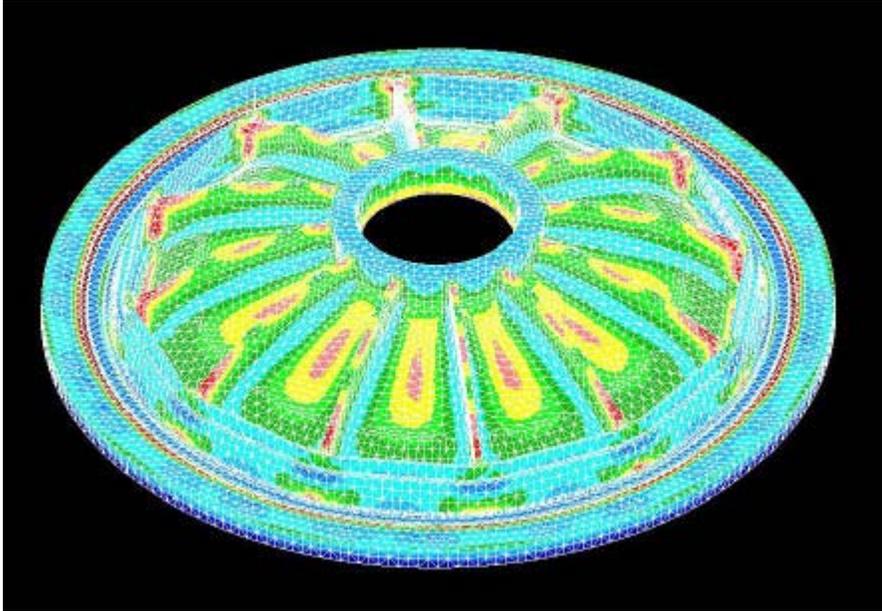
Both source and binary copies of Nastran are available from the Open Channel Foundation for an annual license fee. This was part of the NASA COSMIC Collection distribution from the National Technology Transfer Center.

Competition

Currently there are numerous commercially available FEA products, some of them being able to read the NASTRAN input format although not bearing the name NASTRAN. Many of them are listed on CAE and FEA pages.

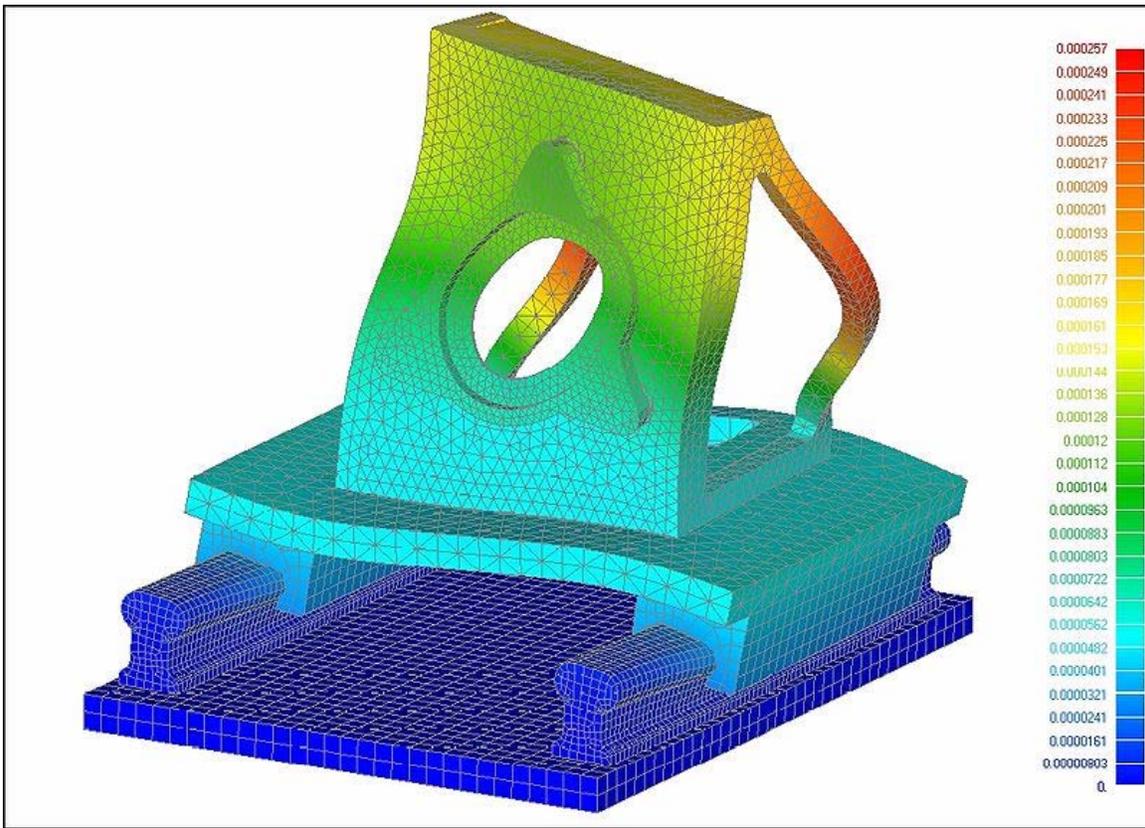
NEi Nastran

NEi Software Products



SpaceShipOne motor bulkhead analyzed in NEi Nastran

Developer(s)	NEi Software
Stable release	V10.0 / 2010
Operating system	Windows XP, Linux
Type	Computer Aided Engineering (CAE) software



Example of NEi Nastran analysis result.

NEi Nastran is an engineering analysis and simulation software product of NEi Software (formerly known as Noran Engineering, Inc.) Based on NASA's Structural Analysis program, the software is a finite element analysis (FEA) solver used to generate solutions for linear and nonlinear stress, dynamics, and heat transfer characteristics of structures and mechanical components. NEi Nastran software is used with all major industry pre and post processors including Femap, a product of Siemens PLM Software, in house brands NEi Fusion, and NEi Works for SolidWorks.

History

The original NASTRAN program came out of NASA's need to develop a common generic structural analysis program that would be used by all of the centers supporting the space program. A specification was written and a contract was awarded to Computer Sciences Corporation for development of NAsa STRuctural ANalysis (NASTRAN) software. NASTRAN was released to NASA in 1968. In addition to NEi Nastran, commercial versions of are also available from Siemens PLM Software (NX Nastran) and others.

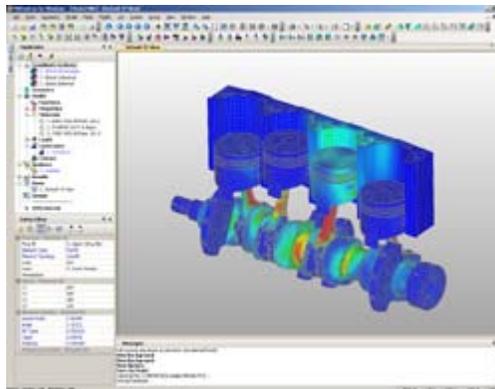
Improvements

In the late 1960s, Finite Element Analysis software was confined to run on expensive mainframe computers and highly trained specialists were needed to apply the program. In this environment, the aerospace industry was the typical user because they had critical projects which could justify the resources FEA demanded. With improvements to the software and wider use of mainframes, FEA technology gradually spread to large corporations that could afford funding the huge investment in hardware, software, and a dedicated FEA staff. Usage spread from primarily aerospace and military applications to the automobile and maritime industries.

The microprocessor revolution and the advent of Personal Computers (PCs) in the 1980s brought tremendous improvements in computing power, significant reductions in computing costs, and the steady development of numerical methods and algorithms. In the mid 1980's, Noran Engineering recognized the long term advantages and impact that the PC hardware revolution could have on the engineering analysis field and embarked on a project to significantly enhance and modernize the original NASTRAN code and port it to PCs.

The first commercial version of NEi Nastran for use on PCs was released in 1990. The new code had a number of changes in architecture and programming language compared to legacy Nastran written originally for mainframes. These differences were intended to take advantage of the dramatic changes in computer hardware taking place and provide the code with key strategic advantages for the new PC platform. For example, since the cost of memory was dramatically reduced it was feasible to perform many operations faster in memory that normally were only done on disk.

Present day



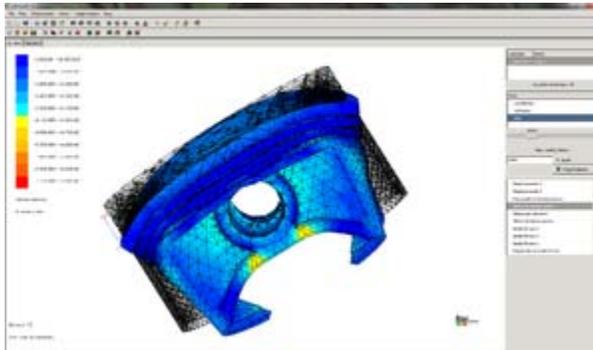
NEi Nastran Screenshot

NEi Nastran V10.0 was released in May 2010. It incorporates over 85 customer driven enhancements including the following additions: nonlinear composite Progressive Ply Failure Analysis (PPFA), concrete material model, direct enforced motion, bolt preload, enhanced rigid element features, visualization support for various entities, automatic

dynamic plots during nonlinear analysis, transparent max/min, and a new look and feel for its Editor tool.

Z88 FEM software

Z88 (Software)



GUI of Z88AuroraV1

Developer(s)	Professor Frank Rieg and team
Stable release	Z88 V13.0, Z88 Aurora V1 / 2010
Operating system	Linux, Windows, UNIX, Mac OS X, 32- &64-bit
Type	Finite element analysis
License	Z88 V13.0 GPL

Z88 is a free (Open Source resp. Freeware) finite element software package available for Microsoft Windows, Mac OS X and Unix/Linux computers. Versions are available for 32-bit and 64-bit operating systems. There are two versions of Z88 available: the Open Source Version Z88 V13 (without preprocessing) and the freeware system Z88Aurora (pre-&post processing). Now Z88 features static calculations in mechanical engineering and building & construction industries. More features like frequency analysis, thermo-mechanical analysis and non-linear behavior will be developed. The complete program is freeware, small, easy to use and build for the use on office computers with different operating systems and due to users feedback constantly being improved. Z88 is coded in ANSI-C.

Development and Contents

Z88 is developed and maintained by Professor Dr.-Ing. Frank Rieg holding the chair of engineering design and CAD at the University of Bayreuth in Germany and his crew. The Open Source Z88 V13 features over 20 different types of finite elements (2D: truss, beam, torus, plane stress elements; 3D: truss, beam, tetrahedron with 4 or 10 nodes, hexahedron with 8 or 20 nodes) a graphical interface and a plot program. Input and output are transparent text files. Several solvers (Cholesky, SICCG, SORCG, Sparse-solver) with multi domain feasibility are included.

Since 2009 a new pre- and post processor called Z88Aurora is developed by a team around Prof. Rieg and was released in June 2010. Z88Aurora uses the technology of Z88 V13.0 combined with an intuitive user prompting (contains boundary conditions, several free FE-mesher, material database with data fitting).

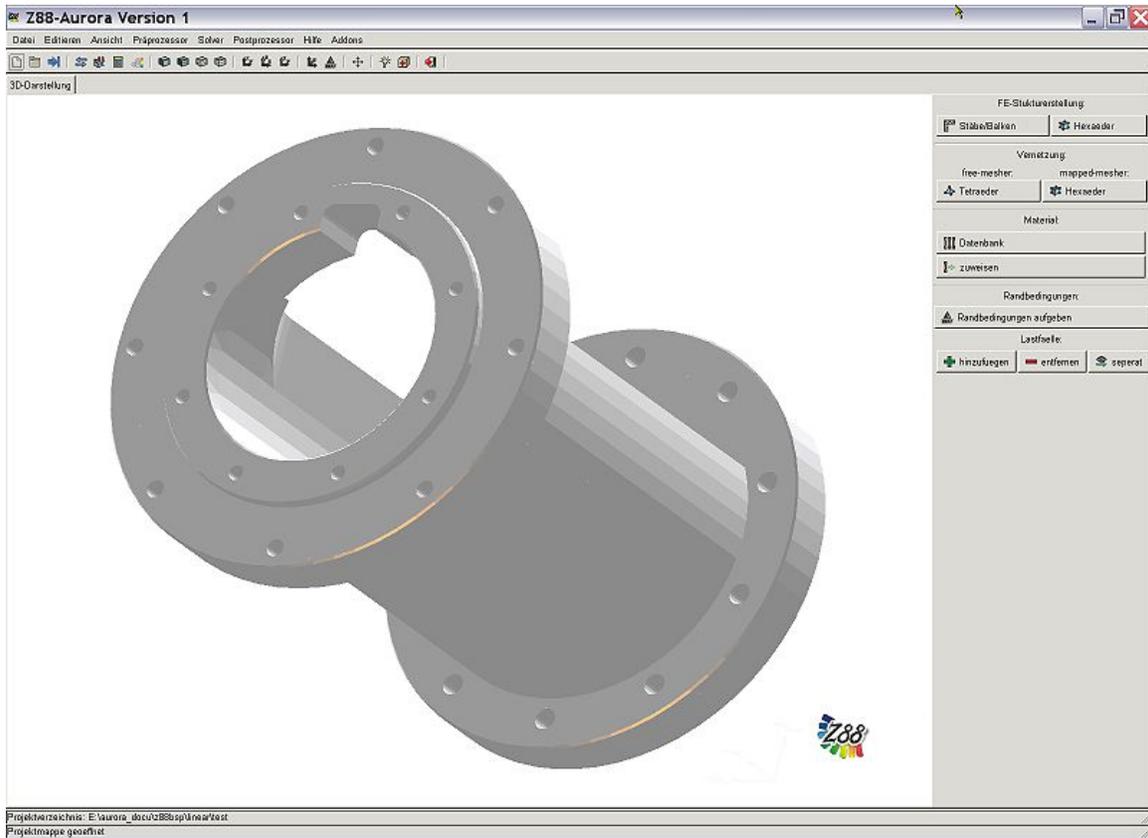
Additional features of Z88Aurora compared with Z88V13:

- shell elements,
- import of fe data: ANSYS, NASTRAN, COSMOS, ABAQUS
- import of geometry data: STL, STEP
- different free and mapped mesher with adjustable options (tetrahedron, hexahedron)
- picking function for loads and constraints (pressure, force, displacements)
- multi core parallel solver
- detailed documentation (user guide, videos, theory guide, ready-to-run examples)

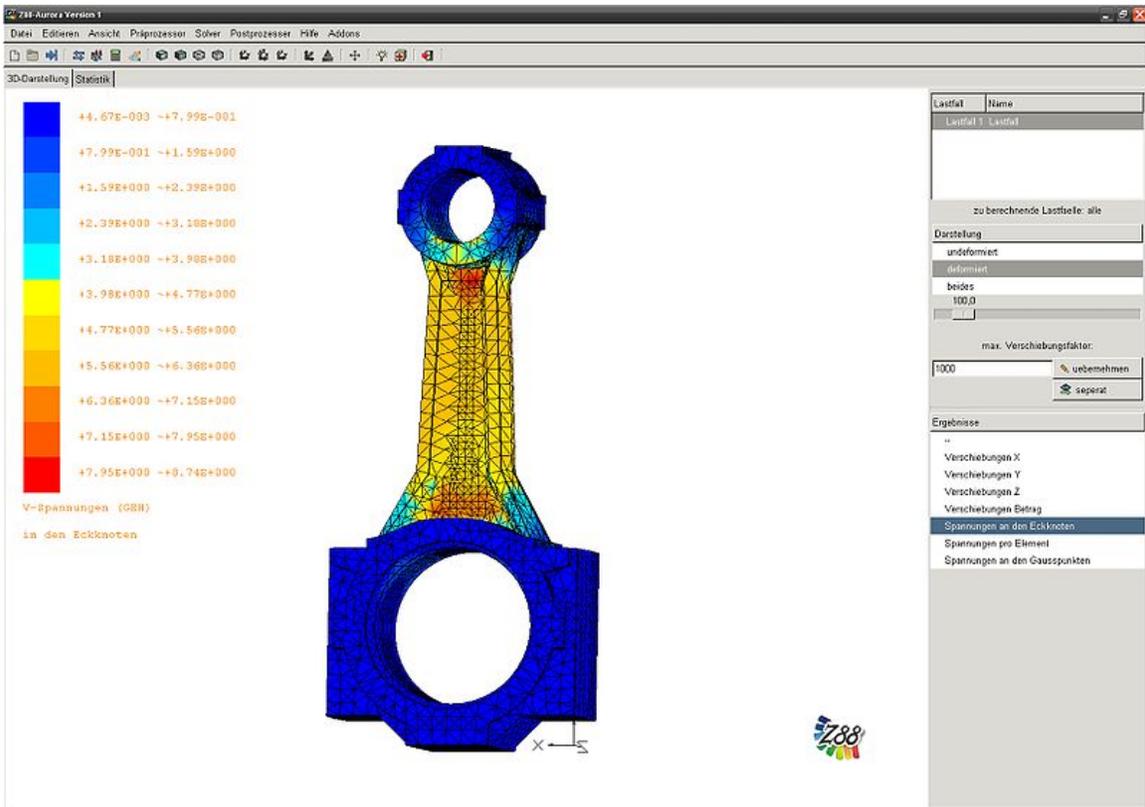
Z88 Aurora snapshots



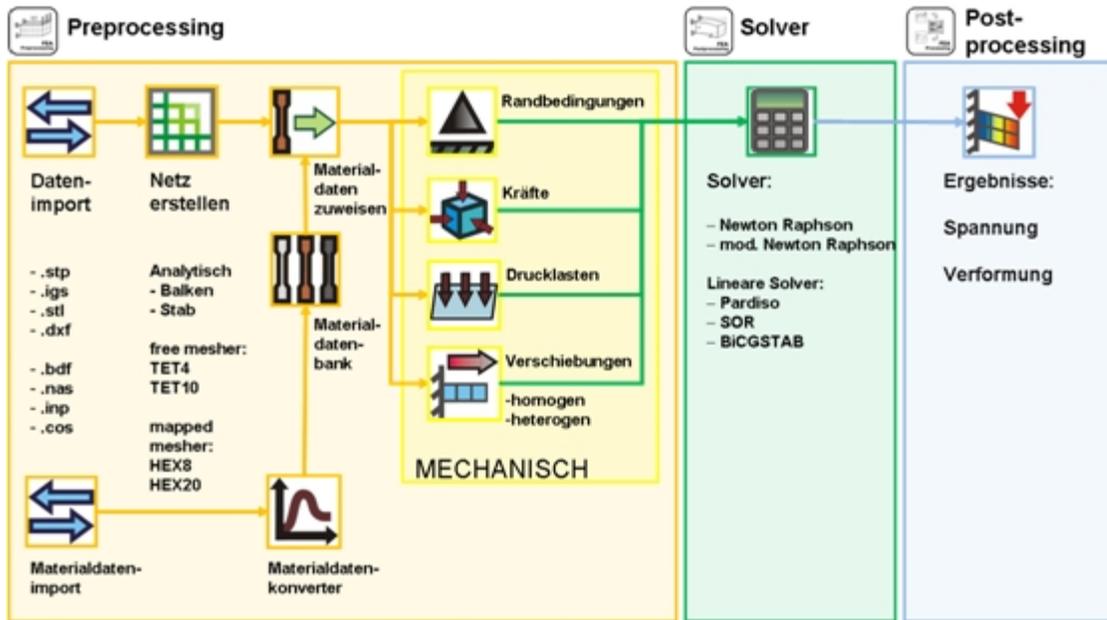
Z88 Aurora



Preprocessing



Postprocessing



Workflow

Use in research and development

Since 1998 the program is used for engineering design lectures at different universities like the TU Dresden, the University of Applied Sciences Ravensburg-Weingarten, the University of Ioannina, the Penn State University, the Universidad de Buenos Aires, the L'Universita Di Cagliari, the University of Maribor and the Zonguldak Karaelmas University.

Application in industry

Due to the open source approach many application use Z88 solvers, the plot programs or the complete software package. Often the possibility of adaption z88 for unique FE-problems is used (calculation of glass plates for architecture, mechanical behavior of wood, computation of pressure vessels, comparison to other fe software etc.) Z88 is used by Boeing Missile and Defense/USA, Teledyne Brown Engineering/USA, Winimac Coil Spring Inc./USA, Double D Design Ltd/New Zealand, RINGSPANN GmbH/Germany, KTR Kupplungstechnik GmbH/Germany, etc.

PLate OPTimizer

PLate OPTimizer, or **PLOP** is a CAD program used by amateur telescope makers to design primary mirror support cells for reflecting telescopes. It was developed by telescope maker David Lewis, first described in 1999, and used to simplify calculations needed in the design of mirror support cells. It was based on Toshimi Taki's 1993 program PLATE, however provides a simplified user interface, thus giving it wide acceptance among makers of large Dobsonian style amateur telescopes, with good support of mirrors as thin as two inches for a diameter of thirty inches.

Comparisons and limitations

A basic mirror cell may be built using minimal calculation and simple materials such as wood and outdoor carpet, with a good example being Dobson's original telescopes. However, as amateurs sought to build larger and thinner mirrors, they found such designs inadequate.

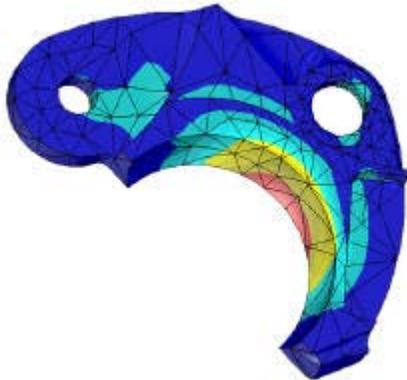
Many amateur telescope makers use cells which are designed via equal area rule calculation, using programs such as David Chandler's public domain program, *Cell*. However, such calculation does not account for mechanical stresses introduced in one part of a telescope mirror by another part, whereas finite element analysis can be used to reduce such stress. Although general finite element analysis programs such as Nastran will work for mirror cells, an advantage of PLOP is that it can be set to ignore deformation that merely results in refocus of a mirror's parabola. PLOP can be used to

calculate floating support points for a mirror's axial (rear) support; however, additional tools are needed to calculate potential error from a mirror's lateral (edge) support.

Mirror cell calculations, whether using PLOP or another program, do not overcome errors introduced by gluing the mirror to its cell, excessive tightening of edge supports, nor impingement of the cell structure onto the mirror as result of differential cooling shrinkage. The significantly more complex calculations arising from the support needs of large honeycomb mirrors and those using active optics systems are outside the design parameters of such programs.

StressCheck

StressCheck



Linear Static Analysis Example

Developer(s)	ESRD, Inc.
Stable release	V9.1 / May 2010
Operating system	Windows XP, Windows Vista
Type	Computer Aided Engineering (CAE) software

StressCheck is a finite element analysis software product developed and supported by ESRD, Inc. of St. Louis, Missouri. It is one of the first commercially available FEA products to utilize the p-version of the finite element method.

History

Development of StressCheck software began shortly after the founding in 1989 of ESRD, Inc. by Dr. Barna Szabó, Dr. Ivo Babuška, and Mr. Kent Myers. The principals have been performing research and development related to p-version finite element analysis for

more than 20 years. Close ties to the Washington University Center for Computational Mechanics facilitates incorporation of the latest research results into StressCheck.

Capabilities

StressCheck is a complete 3D finite element analysis tool with an integrated pre- and post-processor, a suite of analysis modules supporting advanced solutions in elasticity and heat transfer, and utility modules that offer functionality to import CAD models and perform 2D and 3D automatic meshing. Below is an abbreviated summary of current analysis modules and general capabilities.

Pre-Processing

- Fully parametric modeling capability, including parameter-controlled:
 - Geometric dimensioning
 - Meshing parameters
 - Material properties
 - Boundary conditions (loads and constraints)
 - Solution settings
 - Extraction settings
- Geometric (blended) mapping capability for higher-order approximation of geometry
 - This is critical for detailed stress analysis and composite modeling
- Automatic meshing capability in 2D and 3D
- Handmeshing capability in 2D and 3D for improved discretization
- Global-local capability, i.e. importing structural nodal loads from a global model into a local StressCheck detail model
- Standardization via Handbook Solutions and Toolkit FEA
 - Each installation of StressCheck contains a library of pre-built Handbook models to encompass a variety of commonly solved engineering problems.
 - Users may generate organization-specific Handbook solutions for standardization purposes.

Finite Element Solver Features

- Linear Elasticity, including multi-body contact analysis of metallic and composite structures
- Nonlinear Elasticity, including material (i.e. plasticity) and geometric nonlinearities
- Modal/Buckling analysis, including pre-stress buckling
- Steady-State Conduction Heat Transfer, including radiation and convection boundary conditions
- 64-bit Windows batch solver capability for solutions with extraordinary degrees of freedom (DOF)

Post-Processing

- Inherent verification capability for identifying and controlling discretization errors
 - Extract any FEA data of interest (i.e. stress, strain, etc.) and convergence information for that data at any location in the model domain
- Fracture mechanics extractions, including stress intensity factor and J integral computation of separated energy release rates

Interfacing with External Tools

- COM API, allowing the ability to create or load models, solve them, and extract solution data using external programs such as AFGROW, Microsoft Excel and MATLAB
 - Allows for the development of custom applications
 - Optimization programs can interface with parametric modeling capability
 - Automation scripts can be written to update and solve multiple models

Technology

StressCheck uses the p-version of the finite element method. The utilization of the p-version in finite element analysis was pioneered by Dr. Barna Szabó during his tenure at Washington University in St. Louis. The p-version finite element method spans a space of high order polynomials by nodeless basis functions, chosen approximately orthogonal for numerical stability. Since not all interior basis functions need to be present, the p-version finite element method can create a space that contains all polynomials up to a given degree with many fewer degrees of freedom.

In practice, the name p-version means that accuracy is increased by increasing the order of the approximating polynomials (thus, p) rather than decreasing the mesh size, h. Thus, to check for solution convergence by increasing the number of degrees of freedom in a given model, the shape function polynomial level is increased rather than remeshing with more elements, which is the standard FEA tool method. In StressCheck the maximum p-level is set to eight (8).

Application

StressCheck is used in a variety of industries, notably aerospace, and for a range of applications such as aircraft damage tolerance assessment and analysis of composite materials for which high order elements are particularly useful.

STRAND7

	Strand7
Developer(s)	Strand7 Pty. Ltd.
Stable release	2.4 / 2007
Operating system	Windows
Type	Finite Element Analysis Simulator
License	Proprietary

Strand7 is a Finite Element Analysis (FEA) software product developed by the company with the same name.

History

The Strand computer software was first developed by a group of academics from the University of Sydney and the University of New South Wales. Further to this early research work, an independent company called G+D Computing was established in 1988 to develop an FEA program that could be used commercially for industrial applications. Between 1988 and 1996 the company researched, developed and marketed a series of DOS and Unix based FEA programs, most notably its STRAND6 program. In 1996 the company commenced work on a completely new software development specifically for the Windows platform . This product was first released in 2000 and was named Strand7. In 2005 the company also changed its name to Strand7 to better reflect its primary focus .

Application

Some high-profile applications of Strand7 include the optimisation of the "Water Cube" Beijing National Aquatics Center for the 2008 Beijing Olympics, the "Runner" sculpture that was placed on top of Sydney Tower during the 2000 Sydney Olympics and the Terminal 2E roof, Charles de Gaulle Airport.

Analysis Capabilities

Strand7 is most commonly used for the construction and mechanical engineering sectors, but also has seen use in other areas of engineering including aeronautical, marine and mining.

Strand7 includes the following solvers:

- Linear static

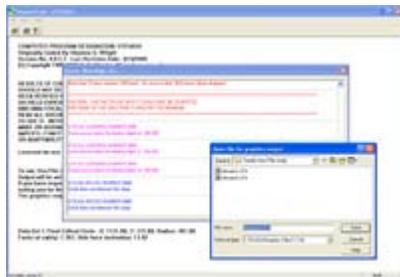
- Natural frequency
- Buckling
- Nonlinear static
- Linear and nonlinear transient dynamic
- Spectral and harmonic response
- Linear and nonlinear steady-state heat transfer
- Linear and nonlinear transient heat transfer

Chapter-7

Geotechnical Engineering Software

UTEXAS

UTEXAS



UTEXAS4 Screen Shot

Original author(s)	Dr. Stephen G. Wright
Stable release	4.0
Operating system	Microsoft Windows
Available in	English
Type	slope stability software
License	Proprietary

UTEXAS is a slope stability analysis program written by Dr. Stephen G. Wright of the University of Texas at Austin. The program is used in the field of civil engineering to analyze levees, earth dams, natural slopes, and anywhere there is concern for mass wasting. UTEXAS finds the factor of safety for the slope and the critical failure surface.

Methodology

UTEXAS uses the limit equilibrium method. The user provides the geometry and shear strength parameters for the slope in question and UTEXAS computes a factor of safety against slope failure. The factor of safety for a candidate failure surface is computed as the forces driving failure along the surface divided by the shear resistance of the soils along the surface.

UTEXAS employs a fast automatic search algorithm to find the failure surface with the lowest factor of safety with respect to shear strength. This is the critical failure surface. Alternatively an arbitrary surface can be entered by the user and UTEXAS can determine the factor of safety associated with it.

The factor of safety for a shear surface is determined using a procedure of slices. Several different procedures exist and the user can choose among them.

Input and output consist of text files. The geologic model is primarily defined using profile lines, which are lines defining the interface between different soil layers. Profile lines are associated with materials, and different materials with different properties can be defined.

Features

The following is a list of some of the more distinct features of UTEXAS:

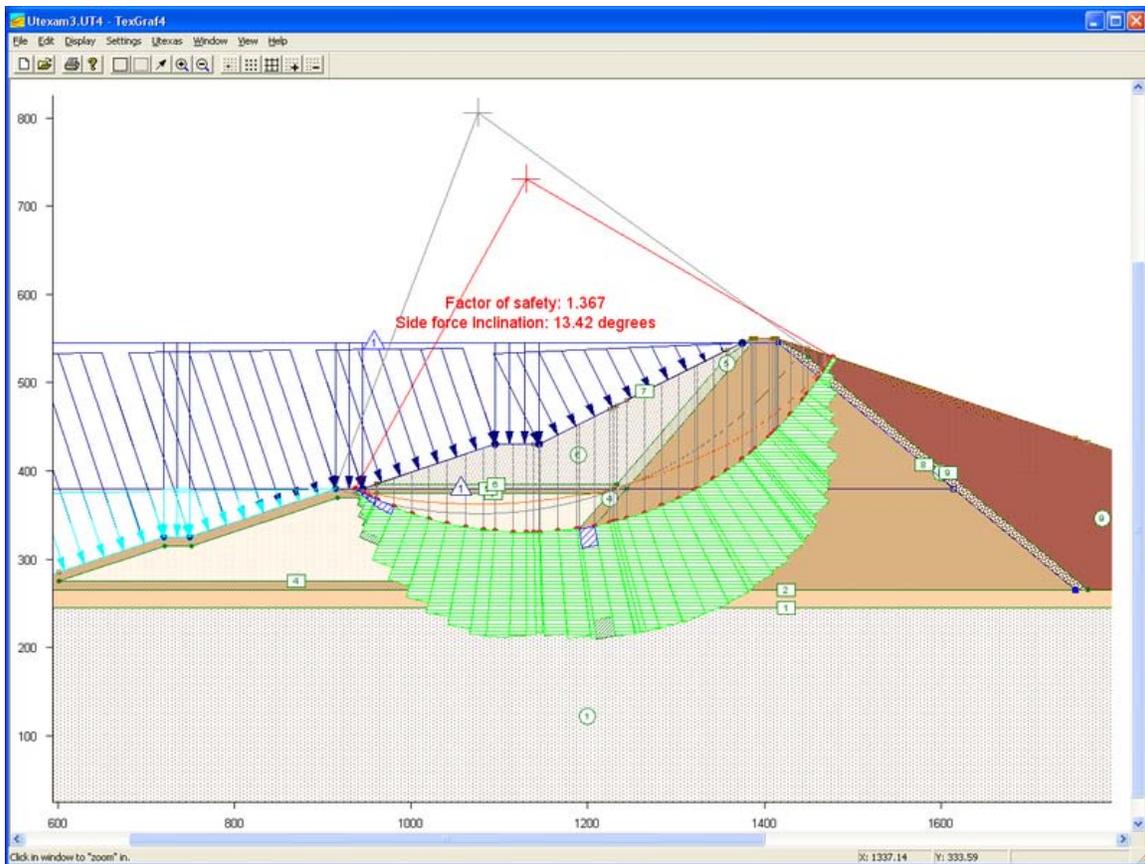
- Automatic Search Grid
- Multi-stage analysis
- Distributed loads
- Line loads
- Non-circular slip surfaces
- Interpolated pore pressures
- Internal soil reinforcement

Version history

- UTEXAS5 - Currently under development
- UTEXAS4 - Released in 1999.
 - Translated into C++ from Fortran.
 - Microsoft Windows interface
 - Accompanied by TexGraf4 program for displaying results.
 - New features: lateral unit weight variation, import of pore water pressure from GMS/SEEP2D, more shear strength options, ability to contour factors of safety, tension cracks
- UTEXAS3 - Released in 1990.
 - First version with multi-stage analysis.
 - Accompanied by GRAPHICS3 program for displaying results.

- New features: Multi-stage analysis
- UTEXAS2 - Released in 1986.
 - First version written for PCs.
 - New features: Multiple limit equilibrium procedures, more ways to define shear strength, internal reinforcement, input data format improvements
- UTEXAS - Released in 1984.
 - Successor to various programs written by Dr. Wright between 1969 and 1984
 - Features: Multiple piezometric surfaces, nonlinear shear strength envelope, simulation of seismic loading, noncircular slip surface search algorithm

Related software



TexGraf4 screen shot

- TexGraf - Reads results from UTEXAS and displays them graphically. Also written by Dr. Wright
- UTEXASED4 - An educational version
- GMS - has a pre and post processor for UTEXAS

Competing software

* Geotechnical Software at the Open Directory Project

SVFlux

	SVFLUX
Developer(s)	SoilVision Systems Ltd.
Operating system	Microsoft Windows
Available in	English
Type	seepage/groundwater software
License	Proprietary

SVFLUX is a finite element seepage analysis program developed by SoilVision Systems Ltd.. The software is designed to analyze both saturated and unsaturated flow through the ground through the solving of Richard's equation. The program is used in the fields of civil engineering and hydrology in order to analyze seepage and groundwater regional flow. The software is used for the calculation of flow rates, pore-water pressures, and pumping rates associated with regional groundwater flow. The software can be coupled with CHEMFLUX in order to calculate diffusion, advection, and decay rates or with SVHEAT in order to calculate thermal gradients and freeze/thaw fronts.

Methodology

SVFLUX makes use of a general finite element solver to solve the Richard's equation for both saturated and unsaturated flow. The finite element solver makes use of automatic mesh generation and automatic mesh refinement in order to aid in problem solution. The software has been used on large projects including the Questa Weathering Study which examined the flow regime through waste rock piles. Several forms of the flow governing equation are implemented in the software which provides greater flexibility in solving unique flow situations.

The user enters geometry, material properties, and analysis constraints through a CAD-type graphical user interface (GUI). The results may also be viewed in the context of a graphical user interface. The geometry is simply entered as regions which may be drawn, pasted in from Excel, or imported from AutoCAD DXF files. The factor of safety for a specific failure surface is computed as the forces driving failure along the surface divided by the shear resistance of the soils along the surface.

A library of benchmark models are distributed with the software.

Free student versions of this software are available for teaching purposes.

Features

The developers of SVFLUX have implemented all of the classic features traditionally found in seepage analysis software as well as an interesting list of new features.

The following is a list of some of the more distinct features of SVFLUX:

- Probabilistic analysis
- Unsaturated analysis with improved convergence
- Coupled climatic boundary conditions and calculation of actual evaporation (AE)
- Automatic mesh generation
- Automatic mesh refinement
- Support for parallel processing
- Large library of example models
- Simple and intuitive graphical user interface

Classic features also supported by the software include:

- Right-click application of boundary conditions and properties
- Help system and tutorial manual
- Solution for saturated and unsaturated flow
- Regional groundwater analysis
- Plotting of flowlines and streamtraces
- Reporting of fluxes

Related software

- SVSLOPE - Limit equilibrium slope stability software
- SVSOLID - Finite element stress / deformation

SVSlope

	SVSLOPE
Developer(s)	SoilVision Systems Ltd.
Operating system	Microsoft Windows
Available in	English
Type	slope stability software
License	Proprietary

SVSLOPE is a slope stability analysis program developed by SoilVision Systems Ltd.. The software is designed to analyze slopes using both the classic "method of slices" as well as newer stress-based methods. The program is used in the field of civil engineering to analyze levees, earth dams, natural slopes, tailings dams, heap leach piles, waste rock piles, and anywhere there is concern for mass wasting. SVSLOPE finds the factor of safety or the probability of failure for the slope. The software makes use of advanced searching methods to determine the critical failure surface.

Methodology

Slope stability problems in geotechnical and geo-environmental engineering involve the solution of equilibrium equations of force and moment. This is traditionally accomplished through traditional method of slices techniques or more progressive stress-based methods. SVSLOPE implements a wide variety of both limit equilibrium methods as well as newer stress-based methods. The classic Bishops, Sarma, Ordinary, Spencers, Morgenstern-Price, GLE, and US Army Corps of Engineers methods are implemented.

The method of slices involves calculating the forces at the base of each slice by the weight of the slice and its height. The newer stress-based techniques use the finite element method in order to calculate more detailed information related to the stresses in the slope. With the finite element-based approach it is possible to represent more complex stress conditions in the slope. The SVSOLID software may be used in order to calculate the finite element stresses in a particular slope.

The user enters geometry, material properties, and analysis constraints (such as searching methodologies) through a CAD-type graphical user interface (GUI). The results may also be viewed in the context of a graphical user interface. The geometry is simply entered as regions which may be drawn, pasted in from Excel, or imported from AutoCAD DXF files. The factor of safety for a specific failure surface is computed as the forces driving failure along the surface divided by the shear resistance of the soils along the surface.

SVSLOPE implements a number of different searching algorithms to identify the critical slip surface. Algorithms for determining both circular and non-circular critical slip surfaces are implemented. Specified slip surfaces can also be employed.

A library of slope stability benchmark models are distributed with the software.

Free student versions of this software are available for teaching purposes.

Features

The developers of SVSLOPE have implemented all of the classic features traditionally found in slope stability software as well as an interesting list of new features.

The following is a list of some of the more distinct features of SVSLOPE:

- Probabilistic analysis
- One-way or two-way sensitivity analysis
- Spatial variability using random fields
- Comprehensive searching algorithms for circular and non-circular slip surfaces
- Unsaturated analysis (coupling with SVFLUX and unsaturated shear strength material properties)
- New finite element based slope stability methods
- Distributed with over 100 example models

Classic features also supported by the software include:

- Distributed loads
- Point loads
- Soil reinforcement
- A variety of ways to represent pore-water pressures
- Non-circular slip surfaces
- Help system and tutorial manual
- Representation of tension cracks
- 17 different soil strength models

Chapter-8

Hydrology Models

Groundwater model

Groundwater models are computer models of groundwater flow systems, and are used by hydrogeologists. Groundwater models are used to simulate and predict aquifer conditions.

Characteristics

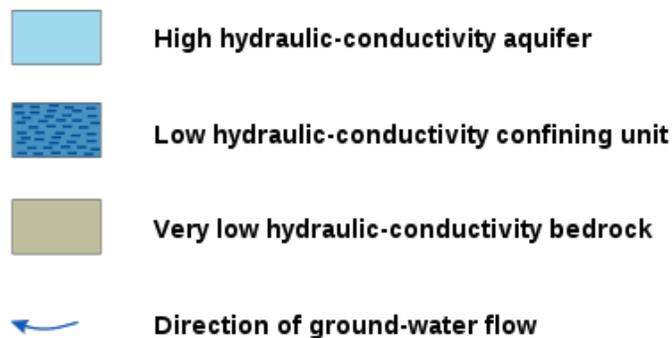
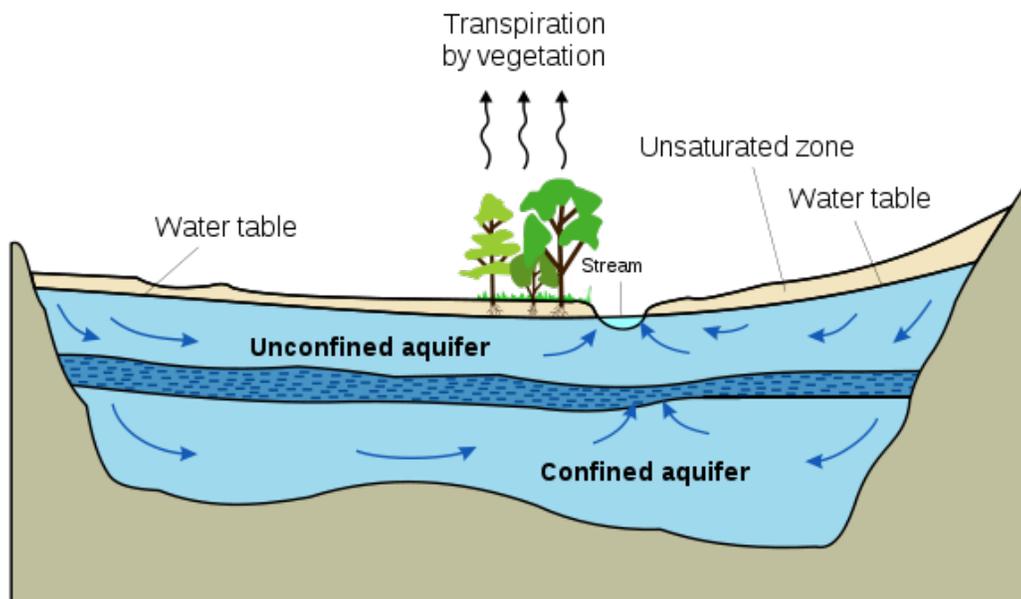


Fig. 1. Typical aquifer cross-section

An unambiguous definition of "groundwater model" is difficult to give, but there are many common characteristics.

A groundwater model may be a scale model or an electric model of a groundwater situation or aquifer. Groundwater models are used to represent the natural groundwater flow in environment. Some groundwater models include (chemical) quality aspects of the groundwater. Such groundwater models try to predict the fate and movement of the chemical in natural, urban or hypothetical scenario.

Groundwater models may be used to predict the effects of hydrological changes (like groundwater abstraction or irrigation developments) on the behavior of the aquifer and are often named groundwater simulation models. Also nowadays the groundwater models are used in various water management plans for urban areas.

As the computations in mathematical groundwater models are based on groundwater flow equations, which are differential equations that can often be solved only by approximate methods using a numerical analysis, these models are also called *mathematical, numerical, or computational groundwater models*.

The mathematical or the numerical models are usually based on the real physics the groundwater flow follows. These mathematical equations are solved using numerical codes such as MODFLOW, ParFlow, HydroGeoSphere etc. Various types of *numerical solutions* like the finite difference method and the finite element method are discussed in "Hydrogeology".

Inputs

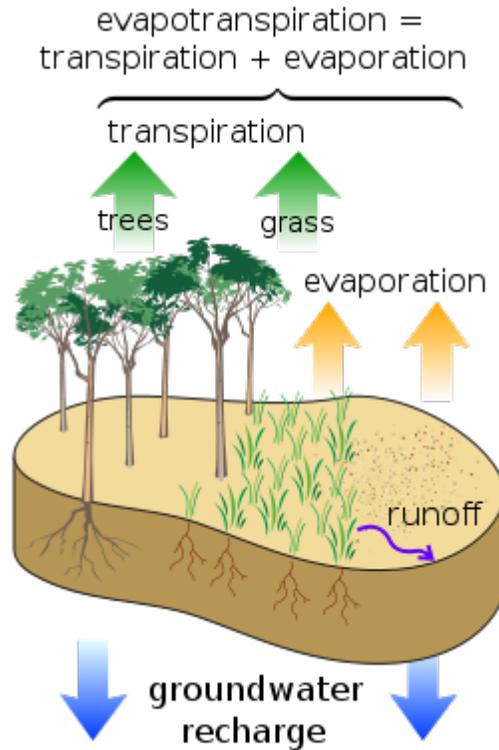
For the calculations one needs inputs like:

- hydrological inputs,
- operational inputs,
- external conditions: initial and boundary conditions,
- (hydraulic) parameters.

The model may have chemical components like water salinity, soil salinity and other quality indicators of water and soil, for which inputs may also be needed.

Hydrological inputs

The hydrological inputs may consist of hydrological data like rainfall, evapotranspiration and surface runoff, which determine the recharge. These inputs may vary both from time to time and from place to place.



Hydrological factors at the soil surface determining the recharge

Operational inputs

The operational inputs concern human interferences with the *water management* like irrigation, drainage, pumping from wells, watertable control, and the operation of retention or infiltration basins, which are often of an hydrological nature. These inputs may also vary in time and space.

Many groundwater models are made for the purpose of assessing the effects hydraulic engineering measures.

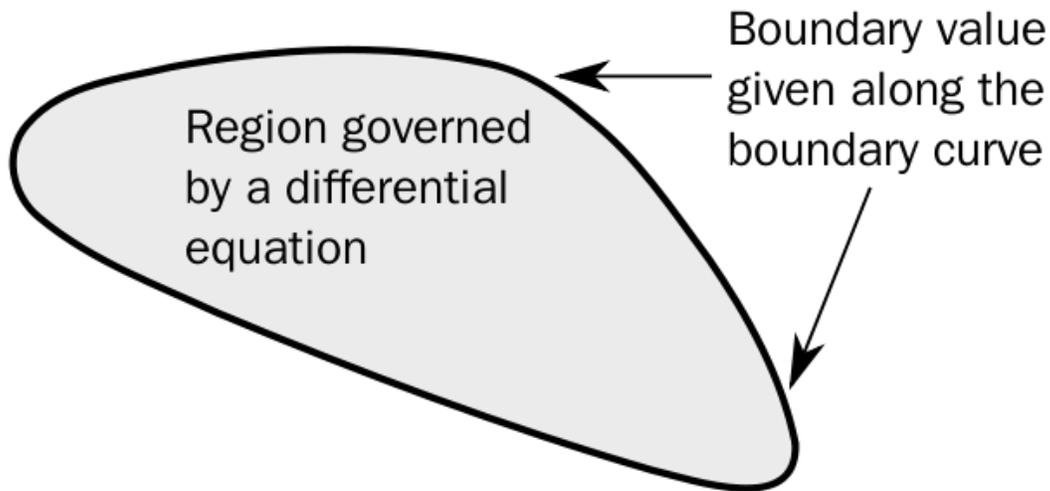


Fig. 2. Boundary conditions

Boundary and initial conditions

Boundary conditions can be related to levels of the water table, artesian pressures, and hydraulic head along the boundaries of the model on the one hand (the *head conditions*), or to groundwater inflows and outflows along the boundaries of the model on the other hand (the *flow conditions*). They may also include quality aspects of the water like salinity.

The *initial conditions* refer to initial values of elements that may increase or decrease in the course of the time *inside* the model domain and they cover largely the same phenomena as the boundary conditions do.

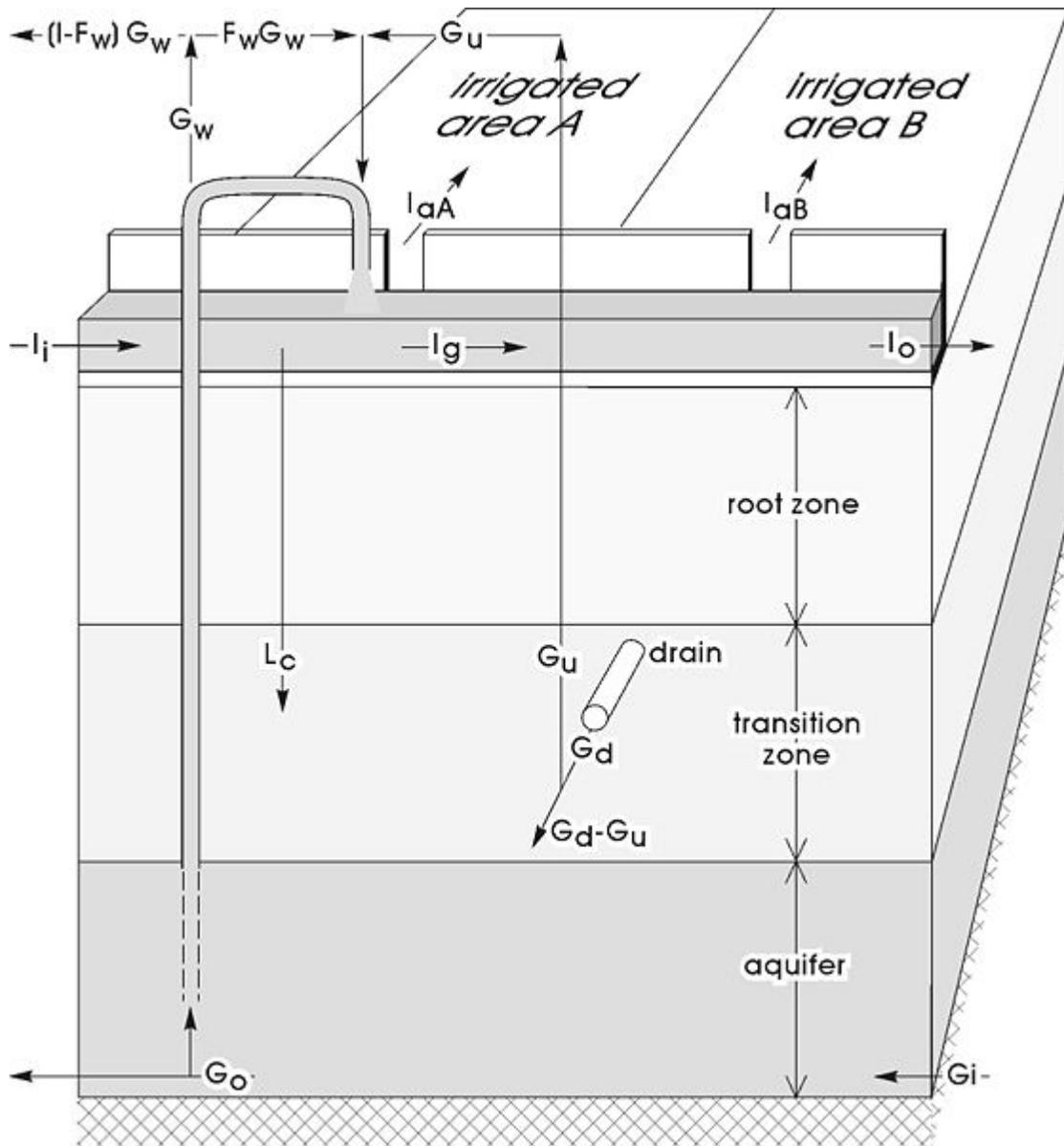


Fig. 3. Example of parameters of an irrigation cum groundwater model

The initial and boundary conditions may vary from place to place. The boundary conditions may be kept either constant or be made variable in time.

Parameters

The parameters usually concern the geometry of and distances in the domain to be modelled and those physical properties of the aquifer that are more or less constant with time but that may be variable in space.

Important parameters are the topography, thicknesses of soil layers and their horizontal/vertical hydraulic conductivity (permeability for water), aquifer transmissivity

and resistance, aquifer porosity and storage coefficient, as well as the capillarity of the unsaturated zone.

Some parameters may be influenced by changes in the groundwater situation, like the thickness of a soil layer that may reduce when the water table drops and/the hydraulic pressure is reduced. This phenomenon is called subsidence. The thickness, in this case, is variable in time and not a parameter proper.

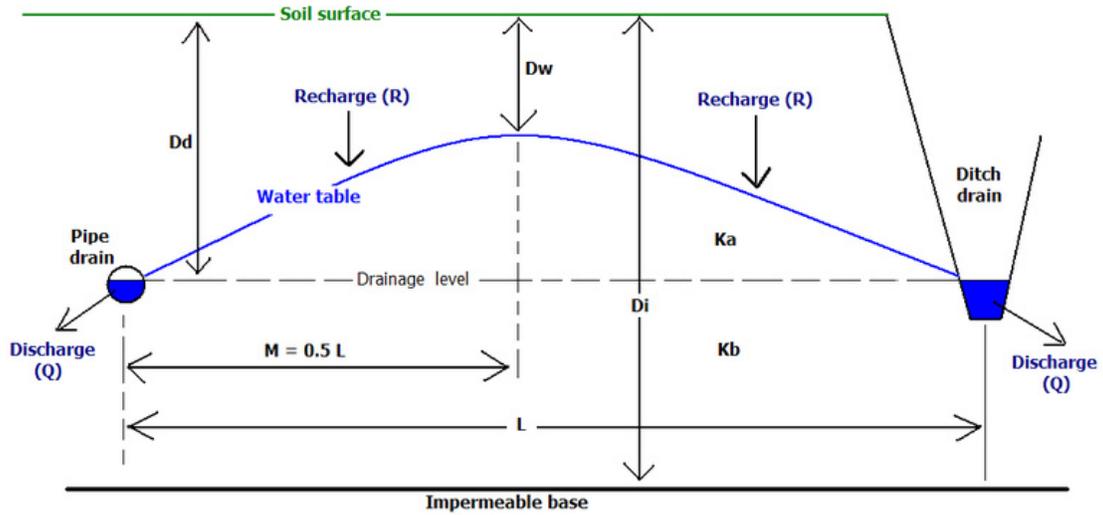
Applicability

The applicability of a groundwater model to a real situation depends on the **accuracy** of the input data and the parameters. Determination of these requires considerable study, like collection of hydrological data (rainfall, evapotranspiration, irrigation, drainage) and determination of the parameters mentioned before including pumping tests. As many parameters are quite variable in space, expert judgment is needed to arrive at representative values.

The models can also be used for the **if-then analysis**: if the value of a parameter is A, then what is the result, and if the value of the parameter is B instead, what is the influence? This analysis may be sufficient to obtain a rough impression of the groundwater behavior, but it can also serve to do a *sensitivity analysis* to answer the question: which factors have a great influence and which have less influence. With such information one may direct the efforts of investigation more to the influential factors.

When sufficient data have been assembled, it is possible to determine some of missing information by **calibration**. This implies that one assumes a range of values for the unknown or doubtful value of a certain parameter and one runs the model repeatedly while comparing results with known corresponding data. For example if salinity figures of the groundwater are available and the value of hydraulic conductivity is uncertain, one assumes a range of conductivities and selects that value of conductivity as "true" that yields salinity results close to the observed values, meaning that the groundwater flow as governed by the hydraulic conductivity is in agreement with the salinity conditions. This procedure is similar to the measurement of the flow in a river or canal by letting very saline water of a known salt concentration drip into the channel and measuring the resulting salt concentration downstream.

Dimensions



Geometry subsurface drainage system by pipes or ditches

D = depth K = hydraulic conductivity L = Drain spacing

Fig 4. Two-dimensional model of subsurface drainage in a vertical plane

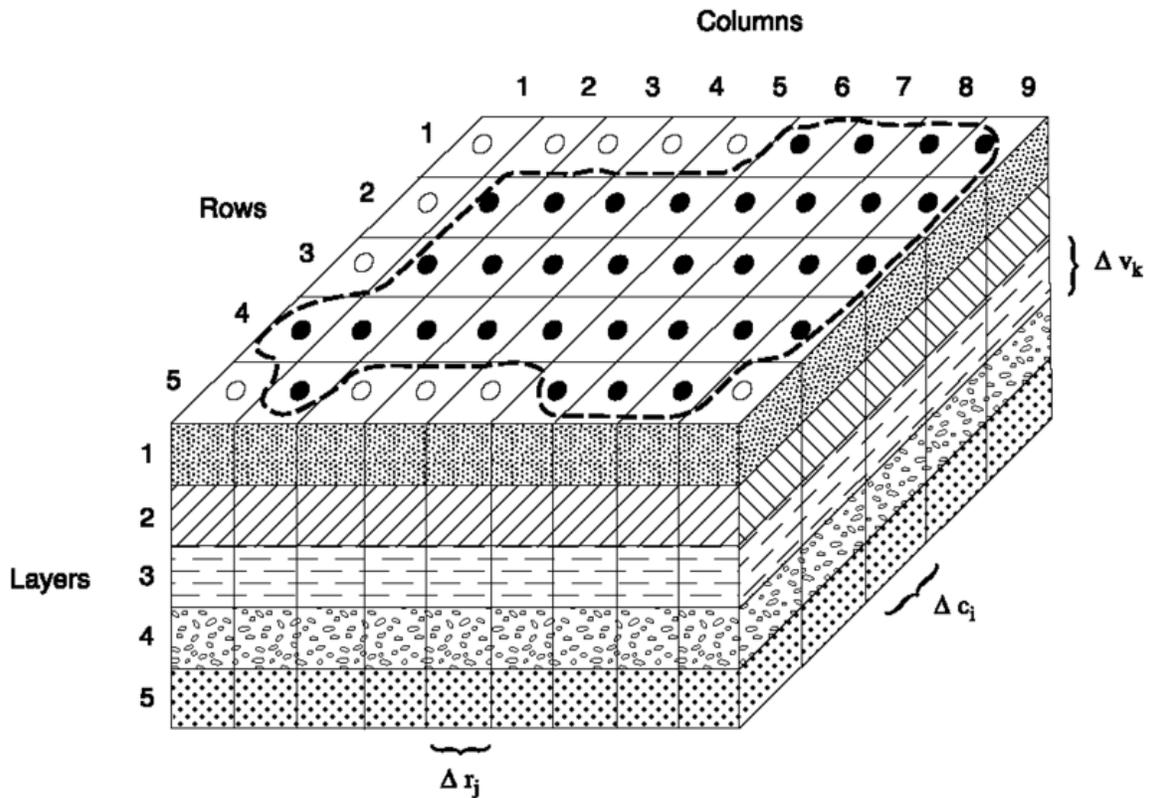


Fig. 5. Three-dimensional grid, Modflow

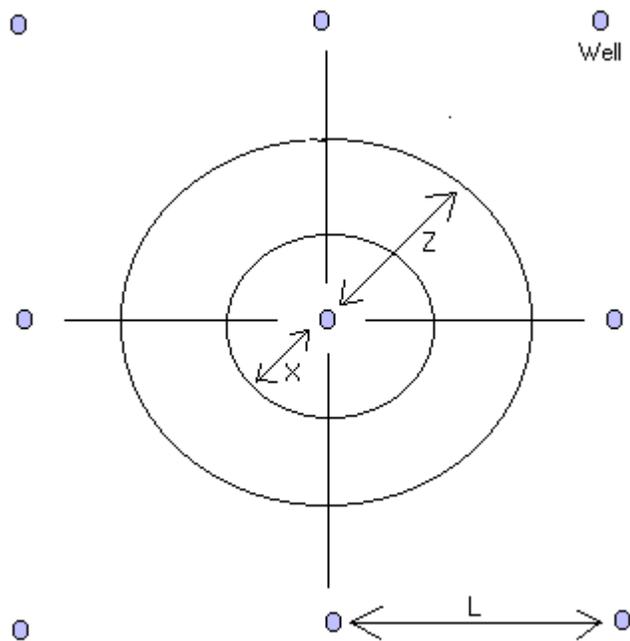


Fig. 6. Map of a radial semi 3-dimensional model consisting of vertical concentric cylinders through which the flow passes radially to the well

Groundwater models can be one dimensional, two dimensional, three dimensional and semi three dimensional. Two and three-dimensional models can take into account the anisotropy of the aquifer with respect to the hydraulic conductivity, i.e. this property may vary in different directions.

One, two and three dimensional

1. One-dimensional models can be used for the vertical flow in a system of parallel horizontal layers.
2. Two-dimensional models apply to a vertical plane while it is assumed that the groundwater conditions repeat themselves in other parallel vertical planes (Fig. 4). Spacing equations of subsurface drains and the groundwater energy balance applied to drainage equations are examples of two-dimensional groundwater models.
3. Three-dimensional models like Modflow require discretization of the entire flow domain. To that end the flow region must be subdivided into smaller elements (or cells), in both horizontal and vertical sense. Within each cell the parameters are maintained constant, but they may vary between the cells (Fig. 5). Using numerical solutions of groundwater flow equations, the flow of groundwater may be found as horizontal, vertical and, more often, as **intermediate**.

Semi three-dimensional

In semi 3-dimensional models the horizontal flow is described by 2-dimensional flow equations (i. e. in horizontal x and y direction). Vertical flows (in z-direction) are described (a) with a 1-dimensional flow equation, or (b) derived from a water balance of horizontal flows converting the excess of horizontally incoming over the horizontally outgoing groundwater into vertical flow under the assumption that water is incompressible.

There are two classes of semi 3-dimensional models:

- *Continuous models* or *radial models* consisting of 2 dimensional submodels in vertical radial planes intersecting each other in one single axis. The flow pattern is repeated in each vertical plane fanning out from the central axis.
- *Discretized models* or *prismatic models* consisting of submodels formed by vertical blocks or prisms for the horizontal flow combined with one or more methods of superposition of the vertical flow.

Continuous radial model

Een example of a non-discretized radial model is the description of groundwater flow moving radially towards a deep well in a network of wells from which water is abstracted. The radial flow passes through a vertical, cylindrical, cross-section representing the hydraulic equipotential of which the surface diminishes in the direction of the axis of intersection of the radial planes where the well is located (Fig.7).

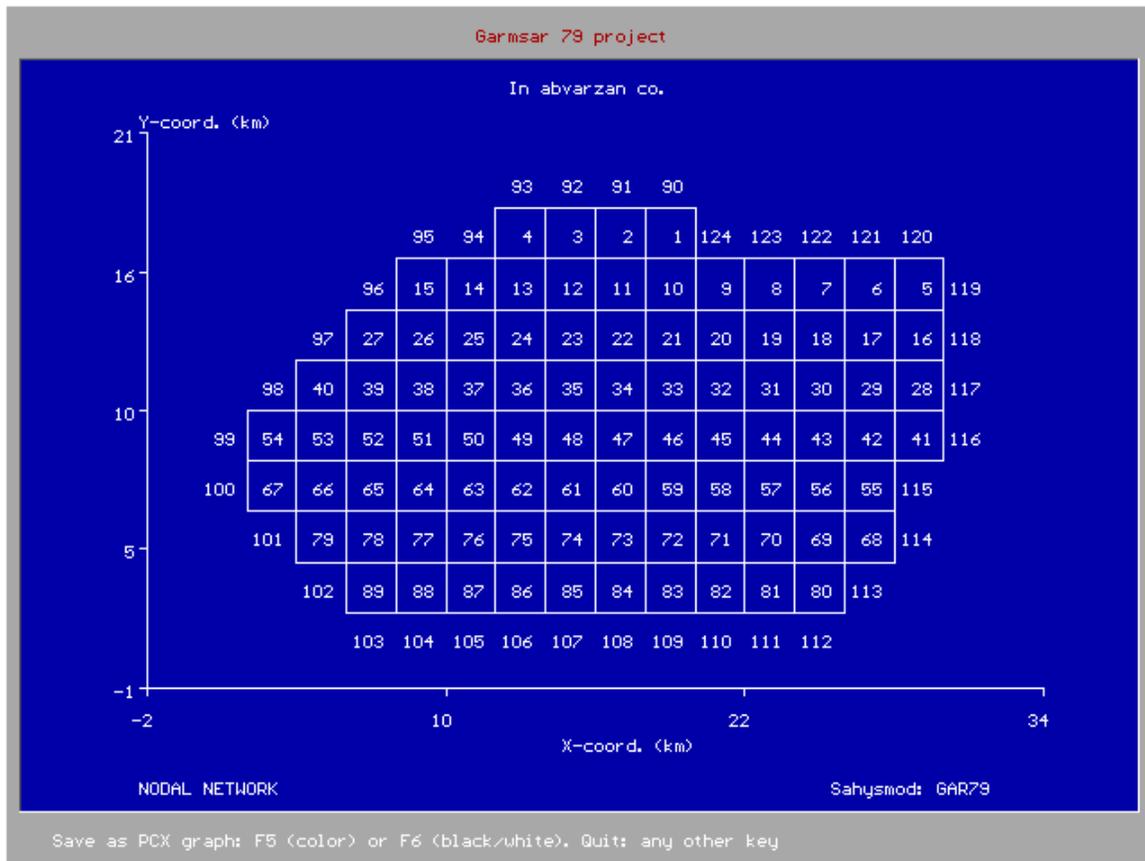


Fig. 7. Map of a two-dimensional grid over an alluvial fan for a prismatic semi 3-dimensional model, SahysMod

Prismatically discretized model

Prismatically discretized models like SahysMod have a grid over the land surface only. The 2-dimensional grid network consists of triangles, squares, rectangles or polygons (Fig. 7). Hence, the flow domain is subdivided into vertical blocks or prisms (Fig. 3). The prisms can be discretized into *horizontal* layers with different characteristics that may also vary between the prisms. The groundwater flow between neighboring prisms is calculated using 2-dimensional horizontal groundwater flow equations. Vertical flows are found by applying one-dimensional flow equations in a vertical sense, or they can be derived from the water balance: excess of horizontal inflow over horizontal outflow (or vice versa) is translated into vertical flow, as demonstrated in Hydrology (agriculture).

In semi 3-dimensional models, intermediate flow between horizontal and vertical is not modelled like in truly 3-dimensional models. Yet, like the truly 3-dimensional models, such models do permit the introduction of horizontal and vertical subsurface drainage systems (Fig. 3).

Semiconfined aquifers with a slowly permeable layer overlying the aquifer (the aquitard) can be included in the model by simulating vertical flow through it under influence of an

overpressure in the aquifer proper relative to the level of the watertable inside or above the aquitard.

Groundwater modeling software & references

- Analytic Element Method
- FEFLOW
- FEHM
- GWFLOW
- HydroGeoSphere
- MicroFEM
- MODFLOW
 - GMS
 - Visual MODFLOW
- SahysMod, Spatial agro-hydro-salinity-aquifer model, online:
- US Geological Survey Water Resources Ground Water Software
- ZOOMQ3D
- PARFLOW

MIKE 11

MIKE 11 is a computer program that simulates flow and water level, water quality and sediment transport in rivers, flood plains, irrigation canals, reservoirs and other inland water bodies. MIKE 11 is a 1 dimensional river model. It was developed by DHI Water • Environment • Health.

MIKE11 has long been known as a software tool with advanced interface facilities. Since the beginning MIKE11 was operated through an efficient interactive menu system with systematic layouts and sequencing of menus. It is within than framework where the latest 'Classic' version of MIKE 11 – version 3.20 was developed.

The new generation of MIKE11 combines the features and experiences from the MIKE 11 'Classic' period, with the powerful Windows based user interface including graphical editing facilities and improved computational speed gained by the full utilization of 32-bit technology.

Modules

The computational core of MIKE 11 is hydrodynamic simulation engine, and this is complemented by a wide range of additional modules and extensions covering almost all conceivable aspects of river modeling.

HD module: it provides fully dynamic solution to the complete nonlinear Saint Venant equations, diffusive wave approximation and kinematic wave approximation, Muskingum method and Muskingum-Cunge method for simplified channel routing. It can automatically adapt to subcritical flow and supercritical flow. It has ability to simulate standard hydraulic structures such as weirs, culverts, bridges, pumps, energy loss and sluice gates.

GIS Extension: it is an extension of ArcMap from ESRI providing features for catchment/river delineation, cross-section and Digital Elevation Model(DEM) data, pollution load estimates, flood visualisation/animation as 2D maps and results presentation/analysis using Temporal Analyst.

RR module: it is rainfall runoff module, including the unit hydrograph method (UHM), a lumped conceptual continuous hydrological model and a monthly soil moisture accounting model. It includes an auto-calibration tool to estimate model parameter based on statistic data of comparison of simulated water levels/discharges and observations.

SO module: it is structure operation module. It simulates operational structures such as sluice gates, weirs, culverts, pumps, bridges with operating strategies.

DB module: it is dam break module. It provides complete facilities for definition of dam geometry, breach development in time and space as well as failure mode.

AUTOCAL module: it is automatic calibration tool. It allows automisation of the calibration process for a wide range of parameters, including rainfall runoff parameters, Manning's number, head loss coefficients, water quality parameters etc.

AD module: it is advection dispersion module. It simulates transport and spreading of conservative pollutants and constituents as well as heat with linear decay.

ST/GST module: it is noncohesive sediment module. It simulates transport, erosion and deposition of non-cohesive and graded noncohesive sediments, including simulations of river morphology.

ACS module: it is cohesive sediment module. It has 3-layer bed description, including quasi-2D erosion.

ECO Lad module: it is ecological modeling. It can simulate BOD/DO, Ammonia, Nitrate, Eutrophication, Heavy metal and Wetlands. It includes standard templates that are well documented and have been used extensively in numerous applications worldwide. Based on predefined process templated, one can develop his/her own templates.

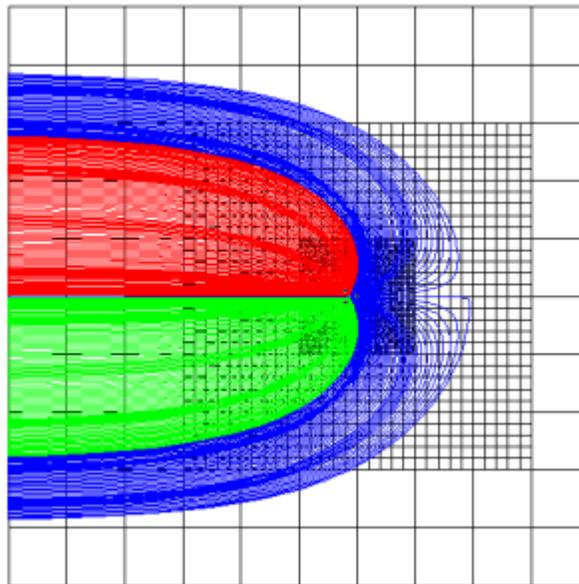
MIKE11 Stratified module: it models vertical density differences such as salinity or temperature in two-layer or multi-layered stratified water bodies.

MIKE11 Real Time module: it is a simulation package and GIS front-end for setting up operational flood forecasting systems. It includes real-time updating and kalman filtering.

Applications

MIKE11 has been used in hundreds of application around the world. Its main application areas are flood analysis and alleviation design, real-time flood forecasting, dam break analysis, optimisation of reservoir and canal gate/structure operations, ecological and water quality assessments in rivers and wetlands, sediment transport and river morphology studies, salinity intrusion in rivers and estuaries.

ZOOMQ3D



ZOOMQ3D grid and particle tracks

ZOOMQ3D is a numerical finite-difference model, which simulates groundwater flow in aquifers. The program is used by hydrogeologists to investigate groundwater resources and to make predictions about possible future changes in their quantity and quality. The code is written in C++, an object-oriented programming language and can compile and run on Windows and Unix operating systems.

Groundwater flow equation

ZOOMQ3D applies a quasi-three dimensional finite-difference approximation to the general three-dimensional governing partial differential groundwater flow equation:

$$\frac{\partial}{\partial x} \left[K_{xx} \frac{\partial \phi}{\partial x} \right] + \frac{\partial}{\partial y} \left[K_{yy} \frac{\partial \phi}{\partial y} \right] + \frac{\partial}{\partial z} \left[K_{zz} \frac{\partial \phi}{\partial z} \right] = S_s \frac{\partial \phi}{\partial t} - q$$

where:

- $\phi(x,y,z,t)$ is the potentiometric head at a point (x,y,z) and time (t) (L)
- K_{xx} , K_{yy} and K_{zz} are the values of hydraulic conductivity along the x , y , and z coordinate axes (LT^{-1})
- q is a volumetric flux per unit volume representing sources and/or sinks of water, where *negative* values are abstractions, and *positive* values are injections (T^{-1}) and,
- S_s is the specific storage of the porous material (L^{-1})

This equation is derived by considering a flow balance for an infinitesimally small volume element located anywhere within a body of saturated aquifer. A number of assumptions underlie this equation. First, the fluid is assumed to be of constant density; this allows the flow balance to be a consequence of mass conservation within the element. Next, the Cartesian coordinate system is aligned with the principal axes of the hydraulic conductivity tensor; this avoids the need for cross derivatives.

A model, based on the above equation, incorporating appropriate boundary and initial conditions, would be truly three-dimensional. ZOOMQ3D takes a simplifying approach to the solution of the three-dimensional equation by recognising that in many aquifers it is possible to identify a layered structure. If the layers are aligned parallel to the horizontal coordinate axes, then the three-dimensional equation can be integrated vertically across the layer to produce an equation which describes the flow within a layer and its interactions with adjacent layers. Such an equation is:

$$\frac{\partial}{\partial x} \left[T_{xx} \frac{\partial h}{\partial x} \right] + \frac{\partial}{\partial y} \left[T_{yy} \frac{\partial h}{\partial y} \right] = S_c \frac{\partial h}{\partial t} - q - L_{\text{above}} + L_{\text{below}}$$

where:

- h is the potentiometric head within a layer (L)
- t , is time (T)
- T_{xx} and T_{yy} are the values of transmissivity along the x and y coordinate axes (L^2T^{-1})
- q is a volumetric flux per unit plan area representing sources and/or sinks of water, where *negative* values are abstractions, and *positive* values are injections (LT^{-1})
- S is the storage coefficient of the porous material (L^0) and,
- L_{above} and L_{below} are leakage rates from layers above and below (LT^{-1})

Model features

Feature	Description
Multiple layers	<p>ZOOMQ3D can incorporate multiple layers of finite difference nodes. The elevation of these layers can vary across the model and the base elevation of one layer can be higher than the top of the layer below it. The separation of model layers simplifies the representation of groundwater systems that contain aquifers separated aquitards. This is because the flow through low permeability layers, which is assumed to be vertical, is represented by the vertical leakage term connecting two finite difference nodes within the upper and lower aquifer.</p>
Local grid refinement	<p>ZOOMQ3D incorporates a mesh refinement procedure which aids the solution of problems related to scale. The density of finite difference nodes can be increased by adding successively finer rectangular grids in discrete areas of the model domain. The mesh can be refined in separate areas and grids can be refined multiple times in the same location in order to zoom into a specific model feature, for example an abstraction borehole or a river reach.</p>
Confined - unconfined conditions	<p>Both confined and unconfined aquifers can be modelled. At confined finite difference nodes transmissivity and storage are independent of groundwater head. At unconfined nodes transmissivity is a function of saturated thickness and the storage term incorporates specific yield. In the top model layer finite difference nodes can be defined as being confined, unconfined or convertible. Convertible nodes switch between unconfined and confined behaviour when the groundwater head rises above its top elevation. In each of the lower model layers, all the nodes must be specified as being either confined or convertible.</p>
Heterogeneity and anisotropy	<p>Finite difference nodes de-water as the groundwater head drops below their base. In this case the node is removed from the matrix of finite difference equations.</p> <p>Models can be heterogeneous and anisotropic. Different hydraulic parameter values can be specified at each finite difference node and hydraulic conductivity may be different in the x and y-directions. It is assumed that the Cartesian coordinate system is aligned with the principal axes of the hydraulic conductivity tensor.</p>
Moving boundaries	<p>Model nodes can de-water and re-wet. Nodes are made inactive when the groundwater level falls below their base and vice versa. The re-wetting of model nodes depends on the groundwater head in adjacent finite difference nodes.</p>

Variable hydraulic conductivity with depth (VKD)	Vertical variations in hydraulic conductivity with depth can be specified within model layers or across model layers by defining VKD profiles. The transmissivity at a node is calculated by integrating the hydraulic conductivity over the vertical saturated thickness of the node.
Recharge	Recharge can vary spatially and temporally. Recharge is always applied to the upper-most active node.
Abstraction wells	Pumped boreholes can be placed at any node within the model domain. Abstraction rates can vary temporally and wells can both abstract water from the aquifer and inject water into it.
Rivers	Dendritic rivers basins are simulated using a series of interconnected river reaches. The hydraulic parameters characterising a reach can vary along the river as can the degree of connection with the aquifer. The transfer of water between the aquifer and rivers is simulated as is the accretion of baseflow along each river branch. Discharges to the river can be specified in any reach, for example to represent a sewage treatment works, and the discharge rate can vary over time. Both fully penetrating and perched rivers can be simulated.
Head-dependent leakage nodes	In addition to rivers, a second head-dependent leakage mechanism is included in ZOOMQ3D. The flow through leakage nodes is proportional to the difference between its elevation and the groundwater head at the finite difference node to which it is connected. Flow can occur in either direction i.e. into or out of the aquifer. Leakage nodes can be used to model spring flows, lakes or estuaries, for example.
Springs	This model feature has been developed to simulate spring flows specifically. The flow out of a spring depends on the transmissivity of the surrounding finite difference nodes. Spring flows are represented by an 'abstraction' which removes water from the aquifer at the location of the spring until the water table falls below the level of the ground surface.
Time discretisation	Simulation time is divided into time-steps, stress periods and blocks. The length of a time-step is equivalent to the length of time between which successive solutions are calculated for the model's state variables. A stress period represents a period of time during which all model stresses remain constant e.g. recharge, groundwater abstraction or discharge to rivers. Stress periods are divided into one or more time-steps. A block is composed of one or more stress periods. The rationale for the use of blocks is predominantly related to the simplification of the organisation of time-variant data, for

example, groundwater abstraction or recharge rates, within input files. The number of stress periods in each block is the same for all blocks within a simulation.

History of development

The groundwater flow model ZOOMQ3D is one of the codes in the ZOOM family of numerical groundwater models which also consists of the advective transport particle tracking code ZOOPT and the distributed recharge model ZOODRM. Each of these models has been developed using object-oriented techniques, a programming approach commonly applied in commercial software development but only relatively recently adopted in numerical modelling for scientific analysis.

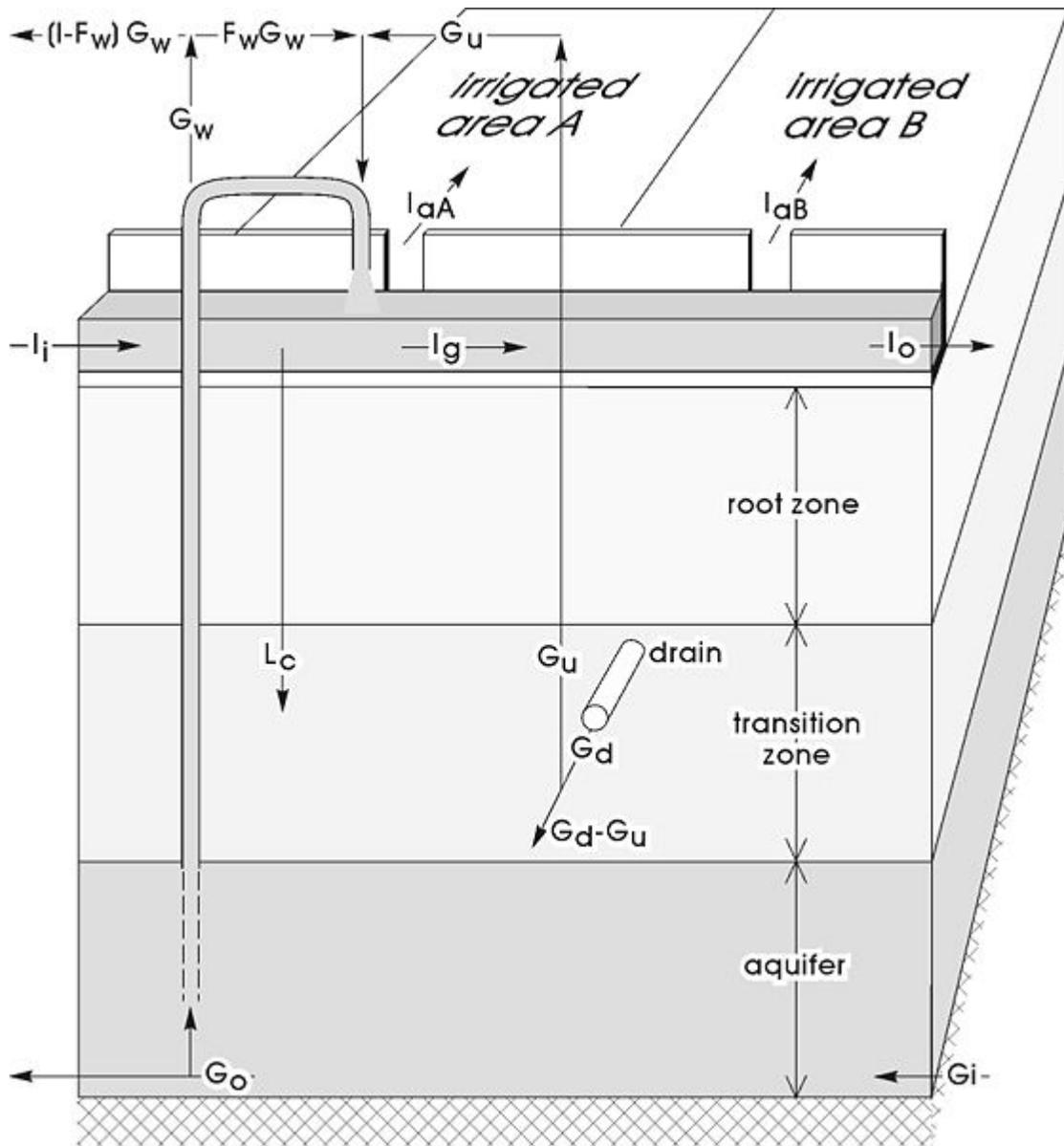
ZOOMQ3D and ZOOPT have been developed through a tri-partite collaboration between the School of Civil Engineering of the University of Birmingham, UK, the British Geological Survey and the Environment Agency of England and Wales. The distributed recharge model ZOODRM has been developed by the British Geological Survey. All models are maintained by the British Geological Survey.

Link to the geological modelling package GSI3D

A link exists between ZOOMQ3D and the geological modelling software GSI3D. The link facilitates the transfer of the structure of a geological model into a ZOOMQ3D groundwater model. After a GSI3D geological model is constructed it can be used to create a hydrogeological model of an aquifer. This is achieved by assigning hydrogeological parameters, such as hydraulic conductivity, to the geological units. Once this has been completed the resulting hydrogeological model can then be converted to a layered ZOOMQ3D groundwater model.

SaltMod

SaltMod is computer program for the prediction of the salinity of soil moisture, groundwater and drainage water, the depth of the watertable, and the drain discharge (hydrology) in irrigated agricultural lands, using different (geo)hydrologic conditions, varying water management options, including the use of ground water for irrigation, and several cropping rotation schedules. The water management options include irrigation, drainage, and the use of subsurface drainage water from pipe drains, ditches or wells for irrigation.



Saltmod components

Soil salinity models

The majority of the computer models available for water and solute transport in the soil (e.g. Swatre, DrainMod) are based on Richard's differential equation for the movement of water in unsaturated soil in combination with a differential salinity dispersion equation. The models require input of soil characteristics like the relation between unsaturated soil moisture content, water tension, hydraulic conductivity and dispersivity. These relations vary to a great extent from place to place and are not easy to measure. The models use short time steps and need at least a daily data base of hydrological phenomena. Altogether this makes model application to a fairly large project the job of a team of specialists with ample facilities.

Simplified salinity model: SaltMod

Rationale

There is a need for a computer program that is easier to operate and that requires a simpler data structure than most currently available models. Therefore, the SaltMod program was designed keeping in mind a relative simplicity of operation to facilitate the use by field technicians, engineers and project planners instead of specialized geo-hydrologists.

It aims at using input data that are generally available, or that can be estimated with reasonable accuracy, or that can be measured with relative ease. Although the calculations are done numerically and have to be repeated many times, the final results can be *checked by hand* using the formulas in the manual.

SaltMod's objective is to predict the long-term hydro-salinity in terms of general trends, not to arrive at exact predictions of how, for example, the situation would be on the first of April in ten years from now.

Further, SaltMod gives the option of the re-use of drainage and well water (e.g. for irrigation) and it can account for farmers' response to waterlogging, soil salinity, water scarcity and over-pumping from the aquifer. Also it offers the possibility to introduce subsurface drainage systems at varying depths and with varying capacity so that they can be optimized. Other features of Saltmod are found in the next section.

Principles

Seasonal approach

The computation method Saltmod is based on seasonal water balances of agricultural lands. Four seasons in one year can be distinguished, e.g. dry, wet, cold, hot, irrigation or fallow seasons. The number of seasons (Ns) can be chosen between a minimum of one and a maximum of four. The larger the number of seasons becomes, the larger is the number of input data required. The duration of each season (Ts) is given in number of months ($0 < T_s < 12$). Day to day water balances are not considered for several reasons:

1. daily inputs would require much information, which may not be readily available;
2. the method is especially developed to predict long term, not day-to-day, trends and predictions for the future are more reliably made on a seasonal (long term) than on a daily (short term) basis, due to the high variability of short term data;
3. even though the precision of the predictions for the future may still not be very high, a lot is gained when the trend is sufficiently clear; for example, it need not be a major constraint to design appropriate soil salinity control measures when a certain salinity level, predicted by Saltmod to occur after 20 years, will in reality occur after 15 or 25 years.

Hydrological data

The method uses seasonal water balance components as input data. These are related to the surface hydrology (like rainfall, evaporation, irrigation, use of drain and well water for irrigation, runoff), and the aquifer hydrology (like upward seepage, natural drainage, pumping from wells). The other water balance components (like downward percolation, upward capillary rise, subsurface drainage) are given as output.

The quantity of drainage water, as an output, is determined by two drainage intensity factors for drainage above and below drain level respectively (to be given with the input data), a drainage reduction factor (to simulate a limited operation of the drainage system), and the height of the water table, resulting from the computed water balance. Variation of the drainage intensity factors and the drainage reduction factor gives the opportunity to *simulate the impact of different drainage options*.

Agricultural data



Irrigated land

The input data on irrigation, evaporation, and surface runoff are to be specified per season for three kinds of agricultural practices, which can be chosen at the discretion of the user:

1. A: irrigated land with crops of group A
2. B: irrigated land with crops of group B
3. U: non-irrigated land with rainfed crops or fallow land

The groups, expressed in fractions of the total area, may consist of combinations of crops or just of a single kind of crop. For example, as the A type crops one may specify the lightly irrigated cultures, and as the B type the more heavily irrigated ones, such as sugarcane and rice. But one can also take A as rice and B as sugarcane, or perhaps trees and orchards. The A, B and/or U crops can be taken differently in different seasons, e.g. A=wheat+barley in winter and A=maize in summer while B=vegetables in winter and B=cotton in summer.

Un-irrigated land can be specified in two ways: (1) as $U=1-A-B$ and (2) as A and/or B

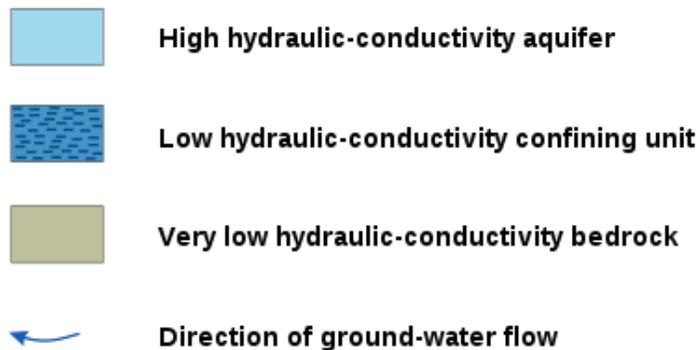
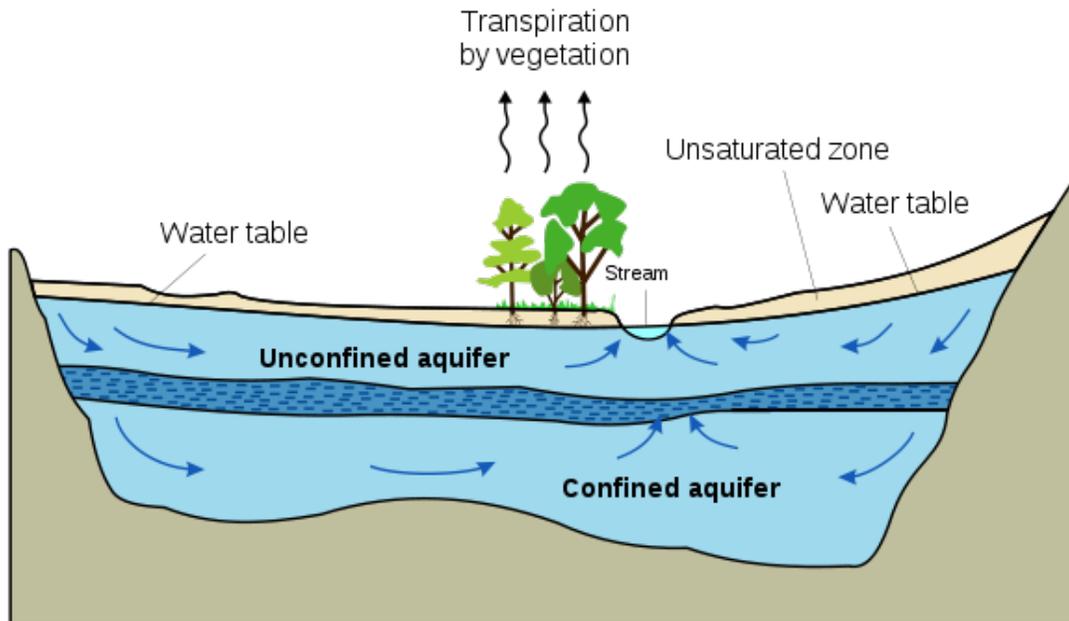
with zero irrigation. A combination can also be made.

Further, a specification must be given of the seasonal rotation of the different land uses over the total area, e.g. full rotation, no rotation at all, or incomplete rotation. This occurs with a rotation index. The rotations are taken over the seasons within the year. To obtain rotations over the years it is advisable to introduce annual input changes.

When a fraction A_1 , B_1 and/or U_1 in the first season differs from fractions are A_2 , B_2 and/or U_2 in the second season, because the irrigation regimes in the seasons differ, the program will detect that a certain rotation occurs. If one wishes to avoid this, one may specify the same fractions in all seasons ($A_2=A_1$, $B_2=B_1$, $U_2=U_1$), but the crops and irrigation quantities may have to be adjusted in proportion.

Cropping rotation schedules vary widely in different parts of the world. Creative combinations of area fractions, rotation indexes, irrigation quantities and annual input changes can accommodate many types of agricultural practices. Variation of the area fractions and/or the rotational schedule gives the opportunity to *simulate the impact of different agricultural practices on the water and salt balance.*

Soil strata



The aquifer can play an important role in soil salinity

Saltmod accepts four different reservoirs, three of which are in the soil profile:

1. a surface reservoir
2. an upper (shallow) soil reservoir or root zone
3. an intermediate soil reservoir or transition zone
4. a deep reservoir or aquifer.

The upper soil reservoir is defined by the soil depth from which water can evaporate or be taken up by plant roots. It can be equal to the rootzone.

The root zone can be saturated, unsaturated, or partly saturated, depending on the water balance. All water movements in this zone are vertical, either upward or downward,

depending on the water balance. (In a future version of Saltmod, the upper soil reservoir may be divided into two equal parts to detect the trend in the vertical salinity distribution.)

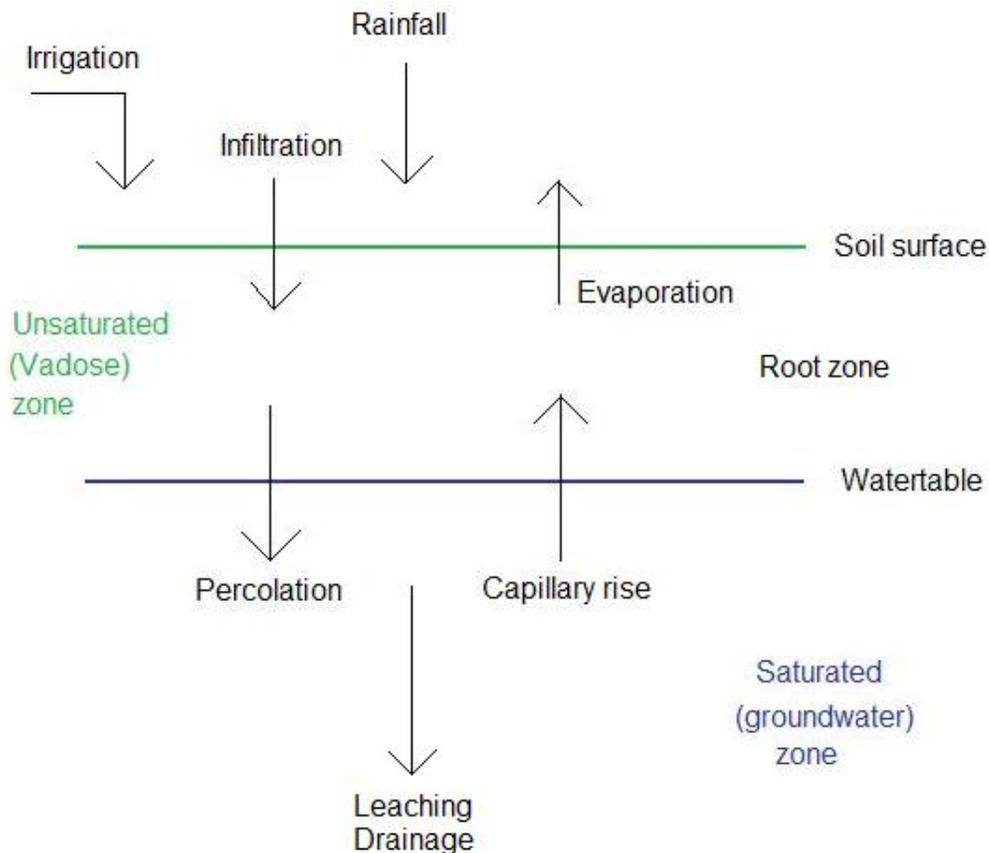
The transition zone can also be saturated, unsaturated or partly saturated. All flows in this zone are vertical, except the flow to subsurface drains.

If a horizontal subsurface drainage system is present, this must be placed in the transition zone, which is then divided into two parts: an upper transition zone (above drain level) and a lower transition zone (below drain level).

If one wishes to distinguish an upper and lower part of the transition zone in the absence of a subsurface drainage system, one may specify in the input data a drainage system with zero intensity.

The aquifer has mainly horizontal flow. Pumped wells, if present, receive their water from the aquifer only.

Water balances



Water balance factors in the top soil

The water balances are calculated for each reservoir separately as shown in Hydrology (agriculture). The excess water leaving one reservoir is converted into incoming water for

the next reservoir.

The three soil reservoirs can be assigned a different thickness and storage coefficients, to be given as input data.

In a particular situation, the transition zone or the aquifer need not be present. Then, it must be given a minimum thickness of 0.1 m.

The depth of the water table, calculated from the water balances, is assumed to be the same for the whole area. If this assumption is not acceptable, the area must be divided into separate units.

Under certain conditions, the height of the water table influences the water balance components. For example a rise of the water table towards the soil surface may lead to an increase of evaporation, surface runoff, and subsurface drainage, or a decrease of percolation losses from canals. This, in turn, leads to a change of the water balance, which again influences the height of the water table, etc.

This chain of reactions is one of the reasons why Saltmod has been developed into a computer program. It takes a number of repeated calculations (iterations) to find the correct equilibrium of the water balance, which would be a tedious job if done by hand. Other reasons are that a computer program facilitates the computations for different water management options over long periods of time (with the aim to simulate their long-term impacts) and for trial runs with varying parameters.

Drains, wells, and re-use

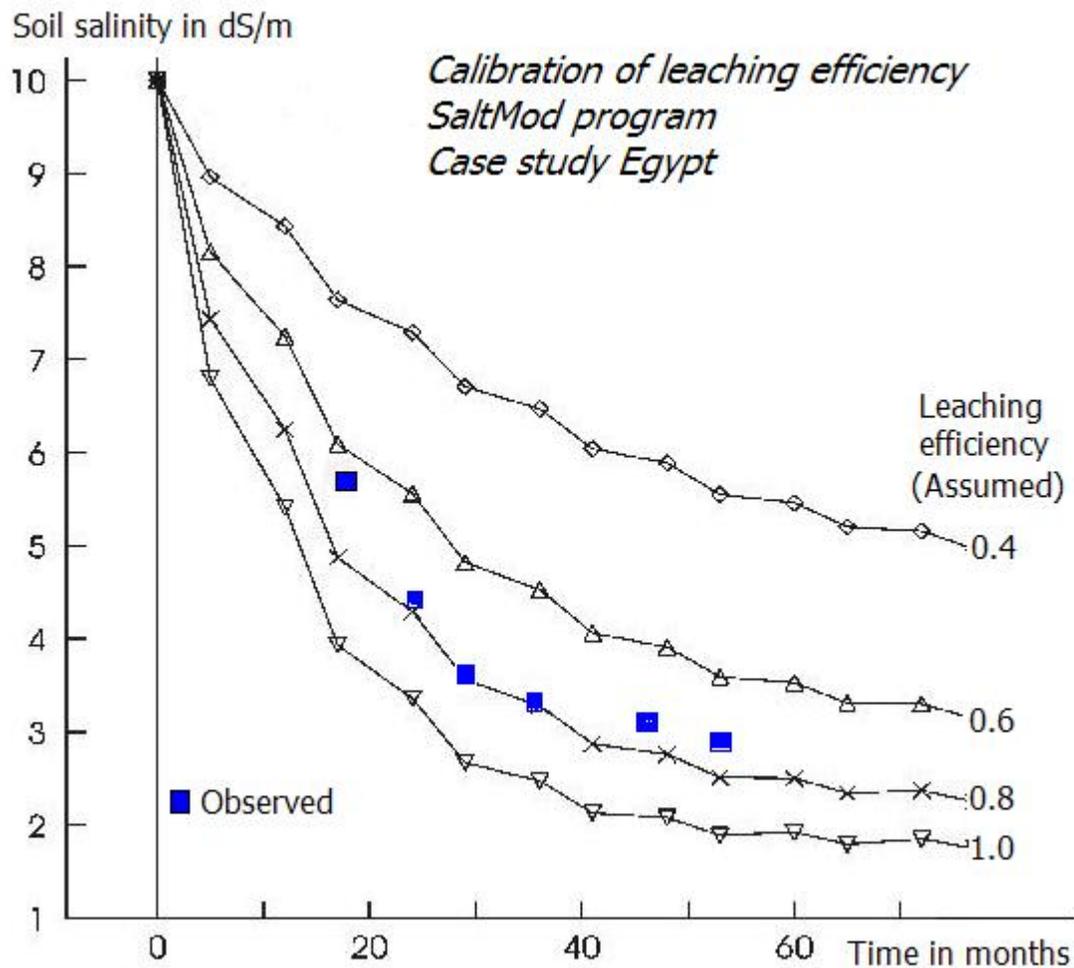
The sub-surface drainage can be accomplished through drains or pumped wells.

The subsurface drains are characterized by drain depth and *drainage capacity factor*. The drains are located in the transition zone. The subsurface drainage facility can be applied to natural or artificial drainage systems. The functioning of an artificial drainage system can be regulated through a *drainage control factor*.

When no drainage system is present, installing drains with zero capacity offers the opportunity to obtain separate water and salt balances for an upper and lower part of the transition zone.

The pumped wells are located in the aquifer. Their functioning is characterized by the well discharge.

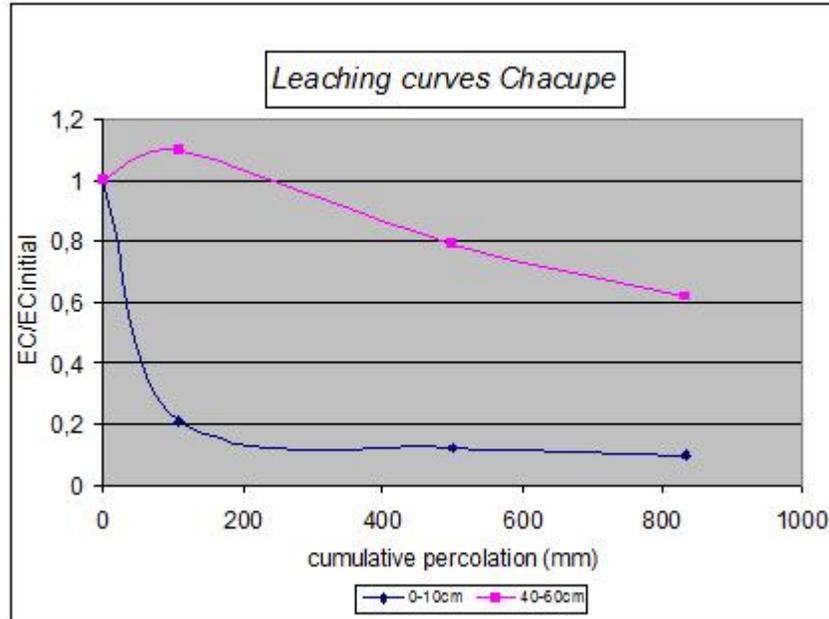
The drain and well water can be used for irrigation through a *re-use factor*. This may have an impact on the salt balance and the irrigation efficiency or sufficiency.



Leaching curves, calibrating leaching efficiency

Salt balances

The salt balances are calculated for each reservoir separately. They are based on their water balances, using the *salt concentrations* of the incoming and outgoing water. Some concentrations must be given as input data, like the initial salt concentrations of the water in the different soil reservoirs, of the irrigation water and of the incoming ground water in the aquifer.



Graphic presentation of soil salinity trends

The concentrations are expressed in terms of electric conductivity (EC in dS/m). When the concentrations are known in terms of g salt/l water, the rule of thumb: 1 g/l → 1.7 dS/m can be used. Usually, salt concentrations of the soil are expressed in E_{Ce}, the electric conductivity of an extract of a saturated soil paste (saturation extract). In Saltmod, the salt concentration is expressed as the EC of the soil moisture when saturated under field conditions. As a rule, one can use the conversion rate EC : E_{Ce} = 2 : 1. Salt concentrations of outgoing water (either from one reservoir into the other or by subsurface drainage) are computed on the basis of salt balances, using different *leaching or salt mixing efficiencies* to be given with the input data. The effects of different leaching efficiencies can be simulated by varying their input value. If drain or well water is used for irrigation, the method computes the salt concentration of the mixed irrigation water in the course of the time and the subsequent impact on the soil and ground water salinities, which again influences the salt concentration of the drain and well water. By varying the fraction of used drain or well water (to be given in the input data), the long term impact of different fractions can be simulated. The dissolution of solid soil minerals or the chemical precipitation of poorly soluble salts is not included in the computation method, but to some extent it can be accounted for through the input data, e.g. by increasing or decreasing the salt concentration of the irrigation water or of the incoming water in the aquifer.

Farmers' responses

If required, farmers' responses to water logging and soil salinity can be automatically accounted for. The method can gradually decrease:

1. the amount of irrigation water applied when the water table becomes shallower;

2. the fraction of irrigated land when the available irrigation water is scarce;
3. the fraction of irrigated land when the soil salinity increases; for this purpose, the salinity is given a stochastic interpretation.

Response (1) is different for ponded (submerged) rice (paddy) and "dry foot" crops. The responses influence the water and salt balances, which, in their turn, slow down the process of water logging and salinization. Ultimately an equilibrium situation will be brought about.

The user can also introduce farmers' responses by manually changing the relevant input data. Perhaps it will be useful first to study the automatic farmers' responses and their effect and thereafter decide what the farmers' responses will be in the view of the user. The responses influence the water and salt balances, which, in their turn, slow down the process of water logging and salinization. Ultimately an equilibrium situation will be brought about.

The user can also introduce farmers' responses by manually changing the relevant input data. Perhaps it will be useful first to study the automatic farmers' responses and their effect and thereafter decide what the farmers' responses will be in the view of the user.

Annual input changes

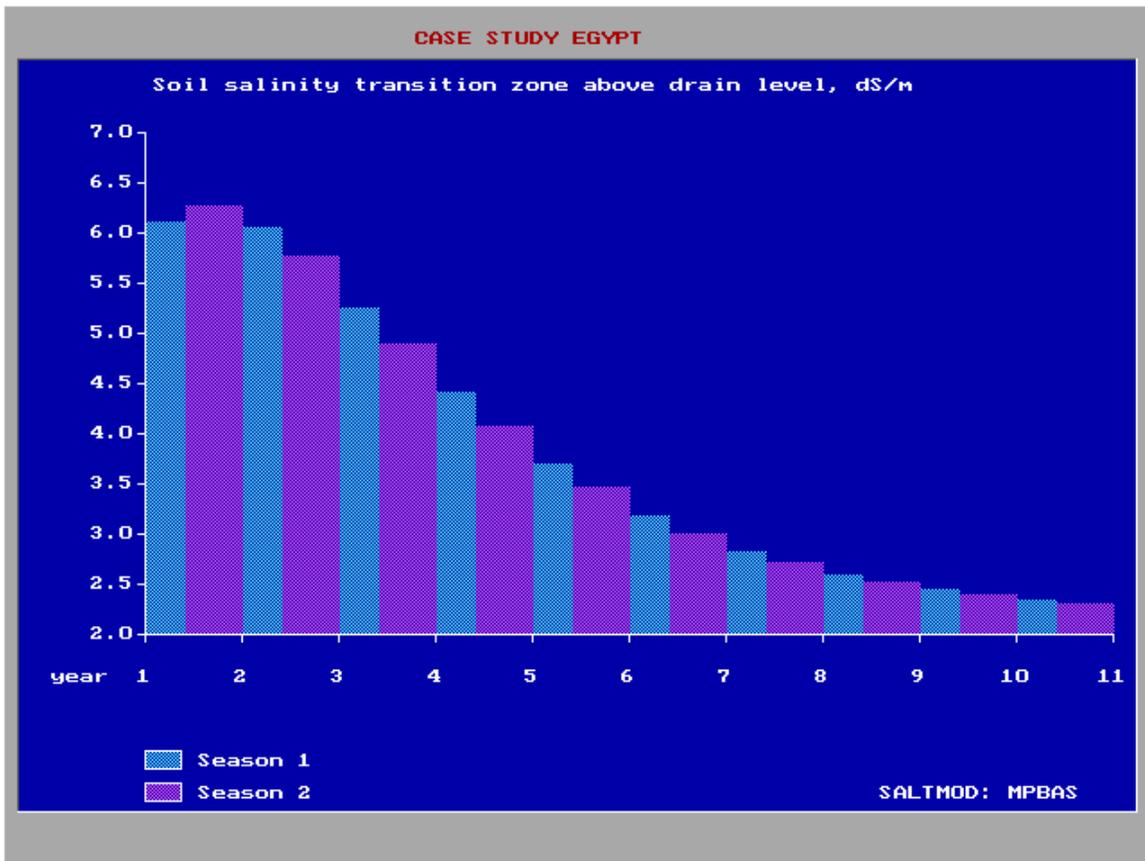
The program may run with fixed input data for the number of years determined by the user. This option can be used to predict future developments based on long-term average input values, e.g. rainfall, as it will be difficult to assess the future values of the input data year by year. The program also offers the possibility to follow historic records with annually changing input values (e.g. rainfall, irrigation, agricultural practices), the calculations must be made year by year. If this possibility is chosen, the program creates transfer files by which the final conditions of the previous year (e.g. water table and salinity) are automatically used as the initial conditions for the subsequent period. This facility makes it possible to use various generated rainfall sequences drawn randomly from a known rainfall probability distribution and obtain a stochastic prediction of the resulting output parameters.

If the computations are made with annual changes, not all input parameters can be changed, notably the thickness of the soil reservoirs and their total porosities as these would cause illogical shifts in the water and salt balances.

Drain depth (D_d , m), soil salinity (C_r , dS/m), field Irrigation efficiency of the group A crops (FaA , -), field irrigation sufficiency of the group A crops (JsA , -), seasonal average depth of the water table (D_w , m), and quantity of drainage water (G_d , mm per season).

Drain Depth D_d	1 s t s e a s o n (s u m m e r)				
	C_r	FaA	JsA	D_w	G_d
0.6	2.7	0.84	0.99	0.37	105
0.8	2.5	0.83	0.98	0.55	112
1.0	2.4	0.82	0.97	0.74	117
1.2	2.2	0.81	0.96	0.93	122
1.4	2.1	0.80	0.95	1.12	127
	2 n d s e a s o n (w i n t e r)				
0.6	2.8	0.86	0.97	0.55	31
0.8	2.7	0.84	0.95	0.74	37
1.0	2.5	0.82	0.93	0.94	45
1.2	2.3	0.81	0.92	1.12	54
1.4	2.2	0.80	0.91	1.31	57

Effects of drain depth, output



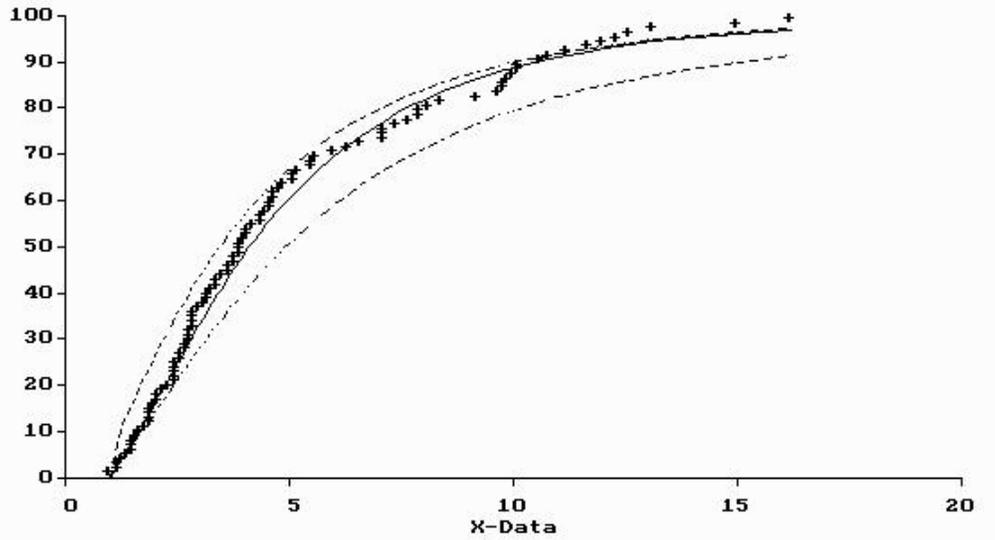
Soil salinity, output

GOHANA DATA, FARMERS' FIELDS, D.P.SHARMA, CSSRI, KARNAL, INDIA

X = soil salinity in ECe (dS/m)

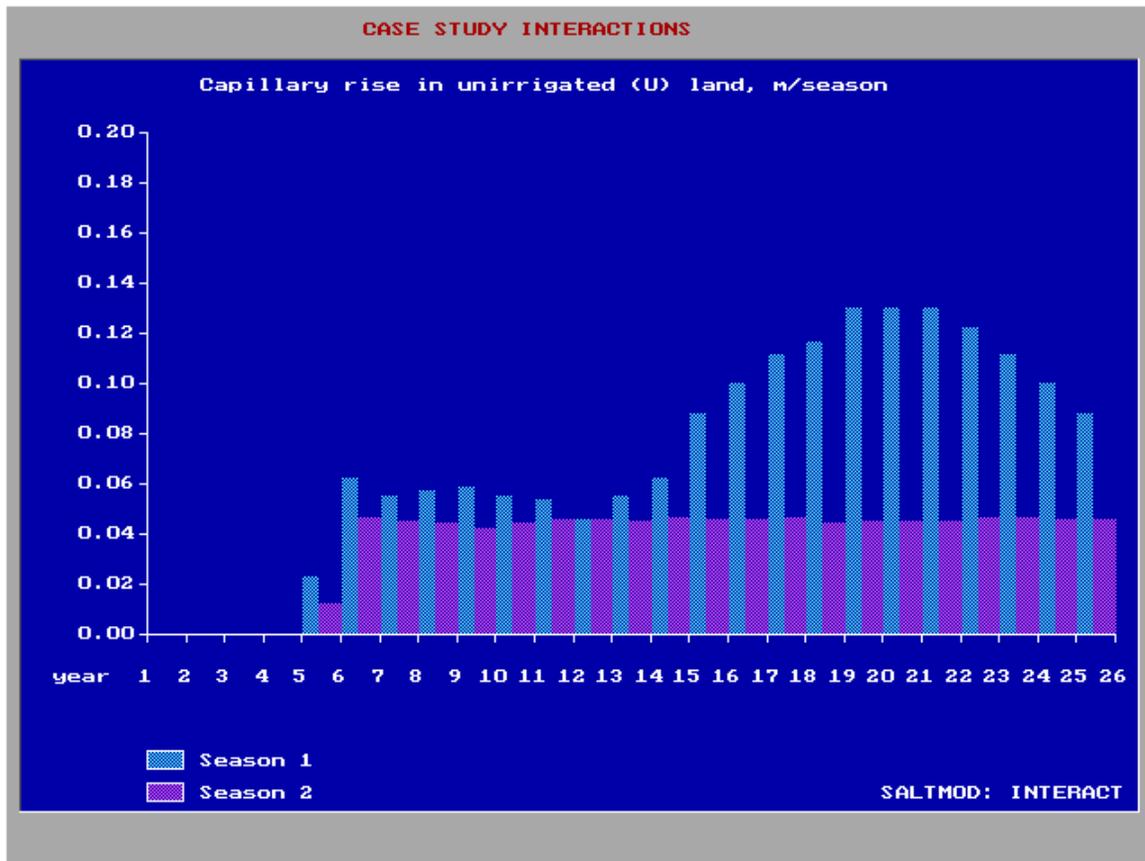
Cumulative frequency (%)

+ observed
— calculated
- - 90% conf.limit



CumFreq: GOHANA2C

Cumulative frequency distribution of soil salinity



Capillary rise, output

Output data

The output of Saltmod is given for each season of any year during any number of years, as specified with the input data. The output data comprise hydrological and salinity aspects.

The data are filed in the form of tables that can be inspected directly or further analyzed with spreadsheet programs.

As the soil salinity is very variable from place to place (figure left) SaltMod includes frequency distributions in the output. The figure was made with the CumFreq program .

The program offers the possibility to develop a multitude of relations between varied input data, resulting outputs and time.

However, as it is not possible to foresee all different uses that may be made, the program offers only a limited number of standard graphics.

The program is designed to make use of spreadsheet programs for the detailed output analysis, in which the relations between various input and output variables can be established according to the scenario developed by the user.

Although the computations need many iterations, all the end results can be *checked by hand* using the equations presented in the manual.