

FINITE ELEMENT OPTION

IBIS Finite Element Front-End

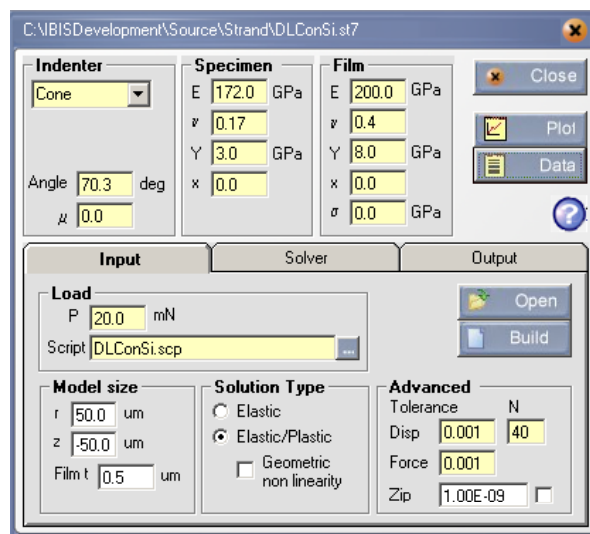
Introduction: One of the most important and valuable tools available to the materials scientist is the availability of numerical solutions to contact problems through the finite element method. Traditionally, such techniques have been the domain of the specialist, but thanks to modern Windows® API functionality, advanced finite element modelling can be performed by the non-specialist using a pre-defined application-specific interface such as that provided by the IBIS software. IBIS allows you to input the indentation parameters (indenter load and geometry, elastic and plastic properties of the specimen, etc). IBIS then automatically creates a mesh with the necessary boundary conditions for full elastic-plastic indentation loading for use with various commercially available finite element analysis programs. .

After the solver has completed, IBIS produces a load-displacement curve which can be directly compared with experimental results, or, depending on other software installed, produce publication-quality contour maps of stresses within the specimen.

***Note:** The IBIS Finite Element Interface requires a separate installation of a finite element analysis program. As of the date of this brochure, Strand7 and ANSYS systems are supported. The FEA interface with IBIS is an accessory to the main IBIS program.

Note: the model built by the IBIS interface to the FEA system is an axis-symmetric model. Pyramidal indenters are converted to an equivalent axis-symmetric cone geometry.

The intention of the IBIS finite element interface is to allow you to create and solve high level finite element models with a user-friendly front-end to an FEA program. As your skills and knowledge of the process increases, you can then use the FEA program directly to create even more advanced models and solutions. The FEA program package can be used in stand-alone mode or with IBIS for indentation work.



IBIS user input of indentation parameters.

FEATURES

- Easy user interface to modeling optimized for indentation analysis.
- Elastic-plastic material properties with strain hardening.
- Optional coating geometry.
- Interfacial friction.
- Residual stress.
- Conical or spherical indenter geometries.
- Variety of output formats.
- **No prior knowledge of finite element analysis required.**
- Potential for many other advanced analyses.

Requirements: IBIS software, Windows 2000/XP/Vista, Strand7 software or ANSYS ED and above.

Finite element User-Input

Input: The IBIS software finite element interface to the FEA program relieves the operator of manual entering all the parameters and building the model. Simply fill in the required material properties as inputs, and the software builds the model automatically.

Indenter: Specifies the indenter type, spherical, conical or cylindrical flat punch.

Elastic modulus: Elastic modulus of the indenter in GPa.

Poisson's ratio: Poisson's ratio of the indenter.

Angle/Radius: Radius in um or cone half-angle in degrees. For a Berkovich or Vickers indenter, enter 70.3 degrees for an equivalent cone.

Friction coefficient: Coefficient of friction for sideways movement between the indenter and contact surface (interfacial friction). Enter 0 for frictionless contact.

Load: in mN to be applied to the indenter.

Script file: contains the fractional load steps which define the way in which load is applied during the solution process. Because the modelling is non-linear, in terms of the expanding area of contact and material plasticity, load is applied in a series of steps. The solution from one load step becomes the initial conditions for the next load step. The load steps can be set using a standard IBIS script file (which may correspond to an experiment) or leave blank for standard steps.

Model Size: These settings define the outer dimensions of the mesh. R and Z define the horizontal and vertical dimensions respectively. These are entered in um units. The objective is to make the outer dimensions of the model large compared to the radius of the circle of contact such that the specimen is regarded as a semi-infinite solid. The IBIS interface defines nodes on the outer edge of the model to be fixed in R and Z. Nodes along the axis of symmetry are fixed in the R direction only.

Solution Type: There are two options: Elastic and Elastic-Plastic. The Elastic option permits a solution to be obtained without material plasticity. Internally, the solution is still non-linear because of the unknown final contact radius. In the Elastic option, the yield stress entry is ignored. The elastic modulus and Poisson's ratio for the specimen are taken as those entered directly. The Elastic-Plastic option includes material plasticity. The IBIS interface constructs a bi-linear stress strain curve based upon the elastic modulus, yield stress and strain hardening coefficient entered. The IBIS interface also specifies the Tresca criterion for plastic flow. The solver applies loads according to the load increments in the script file and adjusts the local modulus of each element so as to satisfy the yield criterion. The load steps are usually specified as a loading and unloading sequence. Permanent deformation is accommodated within the solution so that the unloading response is similar to that which would occur during a real experiment.

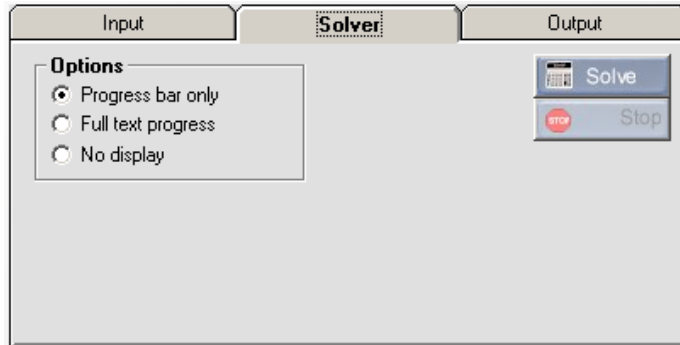
Builds the F.E. mesh by calling ST7 in the background.

Finite element Solver

Solver: Once the model has been built, it can then be solved using the non-linear static solver in the FEA program. The IBIS software either calls the solver in the background or creates a macro file for use within the FEA program itself. The solver can be interrupted if required. Solving the model requires a non-linear iterative approach. Both contact and elastic-plastic behaviour requires that loads be applied in small increments, and mechanical equilibrium achieved before the next load increment is applied.

The solver will produce a series of result step files. Displacements and loads, as well as stresses and strains, are computed for each load step.

The FEA program automatically reduces the load steps to smaller increments to allow for difficult meshes to be solved without user intervention.



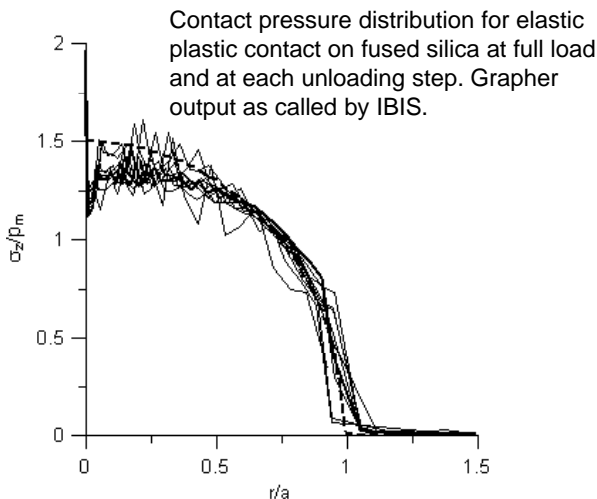
Finite element Output

Output: Once the mesh has been solved, it is required to present the output in a convenient way. The IBIS software interface to FEA program allows a load displacement curve to be created which has the same format as a standard IBIS data file. This allows both finite element and experimental results to be compared numerically side by side.

The IBIS interface can also prepare ASCII text output of stresses and displacements which can then be read into your own preferred graphics program.

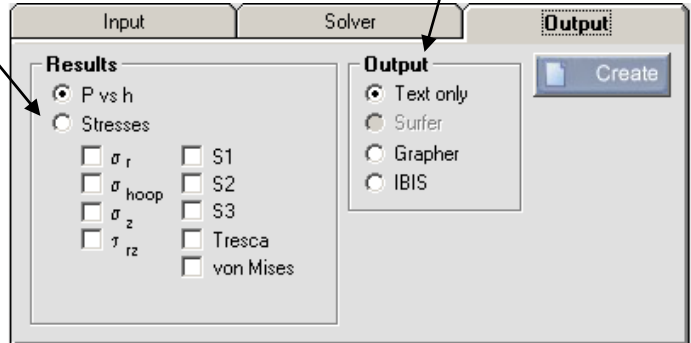
The FEA program itself contains many options for graphical output including stress contouring at each load step, and also animation of the load and unload procedure.

If you have graphing software from Golden Software Inc, then IBIS can directly call either Surfer® or Grapher® and create publication-quality figures in vector graphics format.

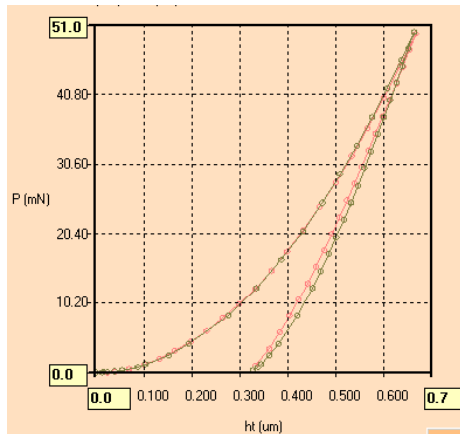


Allows convenient selection of stresses to be included in output.

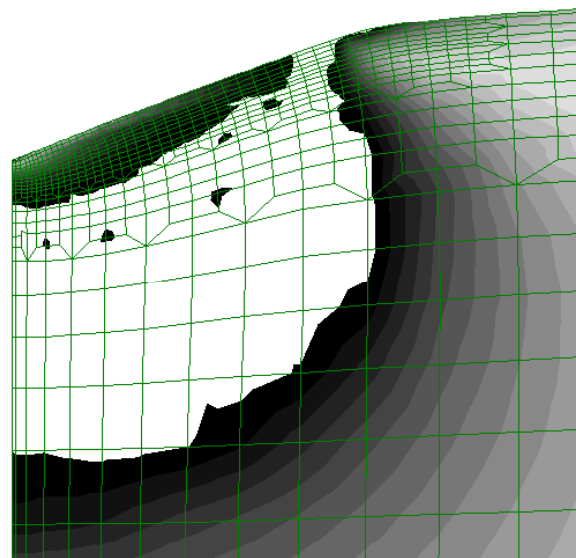
Variety of output formats.



IBIS FE output panel.



Comparison of experiment (Berkovich) and finite element (70.3 cone) results for load displacement curve in IBIS format for elastic-plastic:



Plastic zone at full load for elastic plastic contact on fused silica as generated by FEA graphical output capabilities - bitmap format.

Fischer-Cripps Laboratories Pty. Limited

P.O. Box 9, Forestville NSW 2087 Australia.
www.ibisonline.com.au