EUROPEAN UNION
EUROPEAN REGIONAL DEVELOPMENT FUND
INVESTMENT IN YOUR FUTURE

PragTic

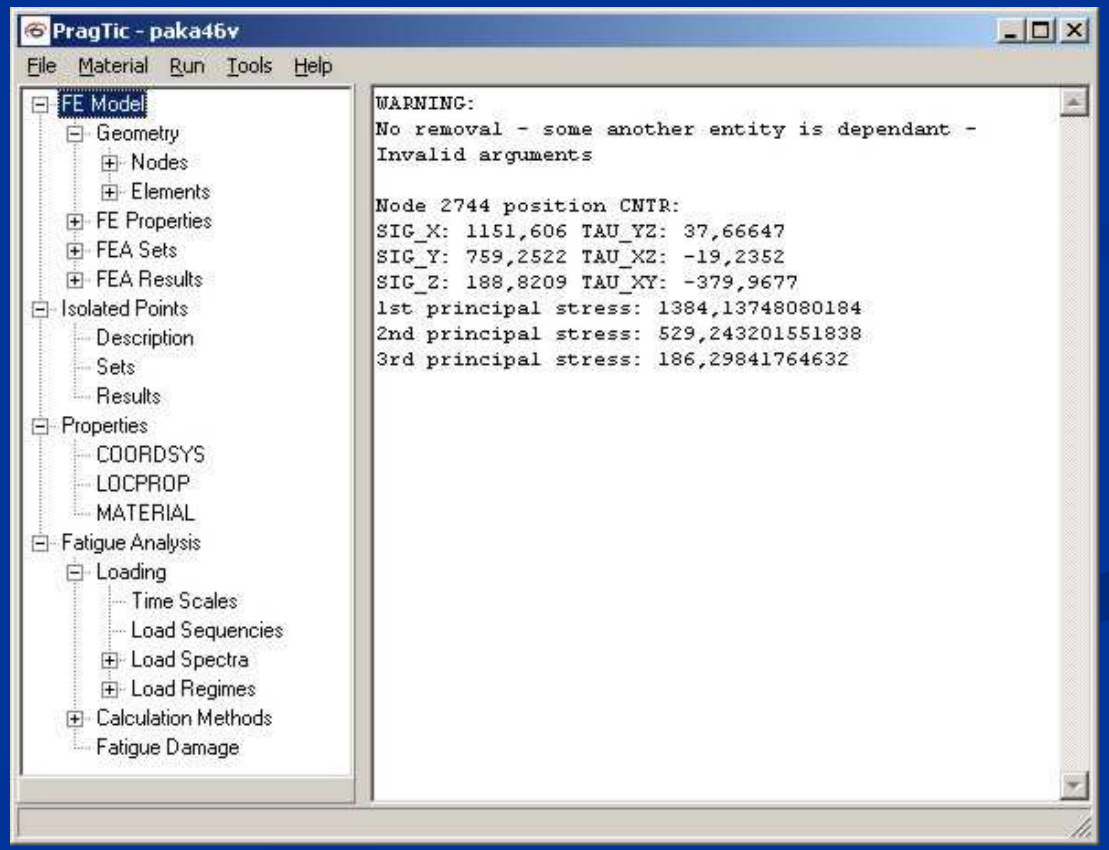
Use of Finite Element Analysis Data in Fatigue Analyses

Jan Papuga

The organisation of the meeting is partially financed from OPPI programme
Czech Technology Airspace Platform, reg. number SPTP 01/004

Software PragTic

- Started as a support tool for preparation of my PhD thesis
- Present focus: research tool usable for fatigue computation
- Freeware (www.pragtic.com)
- MS Windows based
- English interface only
- v.0.2beta - v.0.2betaD development sponsored by Evektor
- Previous versions sponsored by CTU in Prague

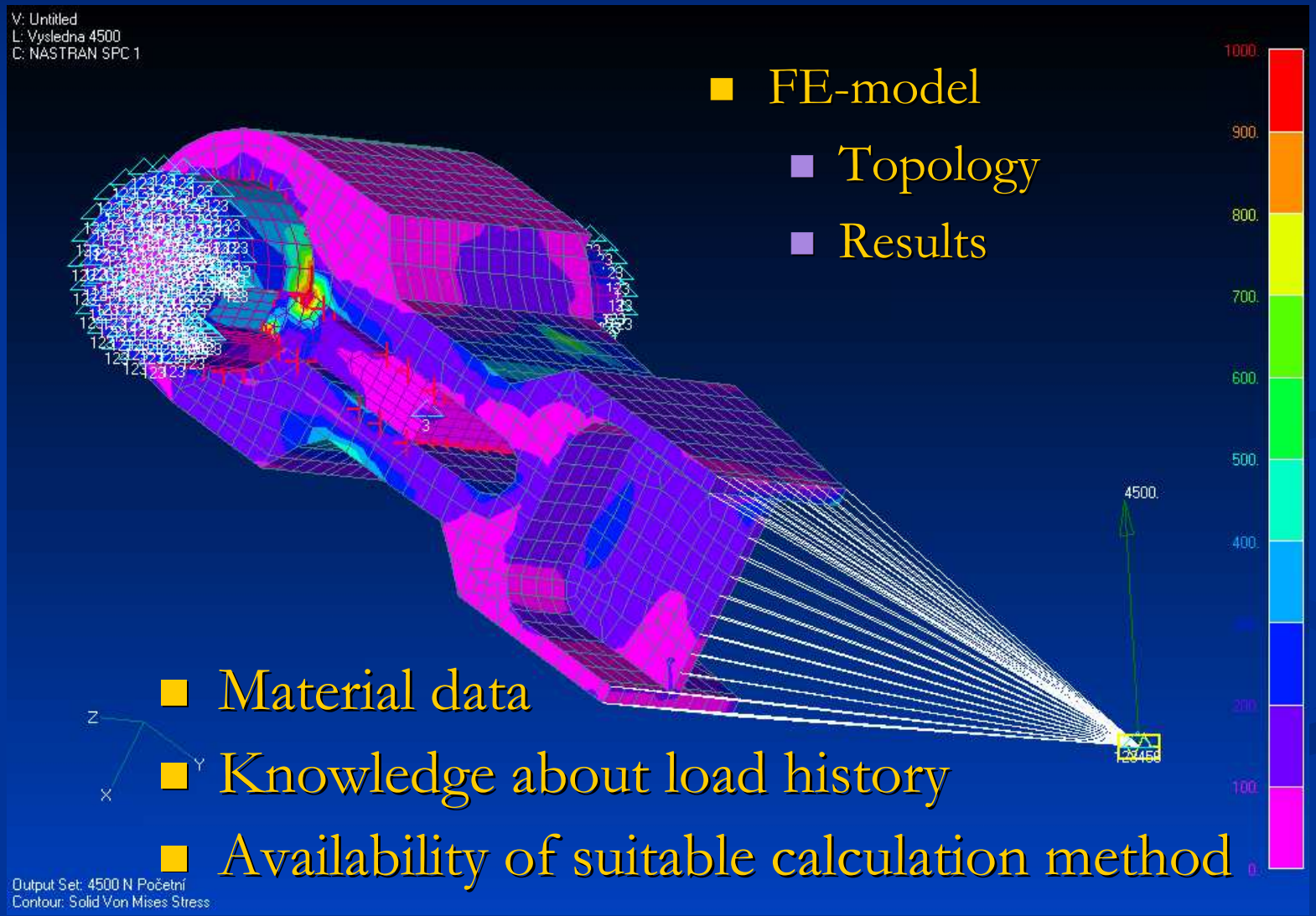


PragTic Features

- Works on data
  - of FE-model
  - at isolated points with no relation to any FE-model (strain gauges)
- Own database of all data processed
- Preparation and analysis of loading by more load channels
- Easy to use graphical interface
- Complex setup of calculation methods
- Unique coverage of multiaxial methods
- Approx. 20 high-cycle and 10 low-cycle fatigue computation methods

Fatigue Calculation on FE-Model

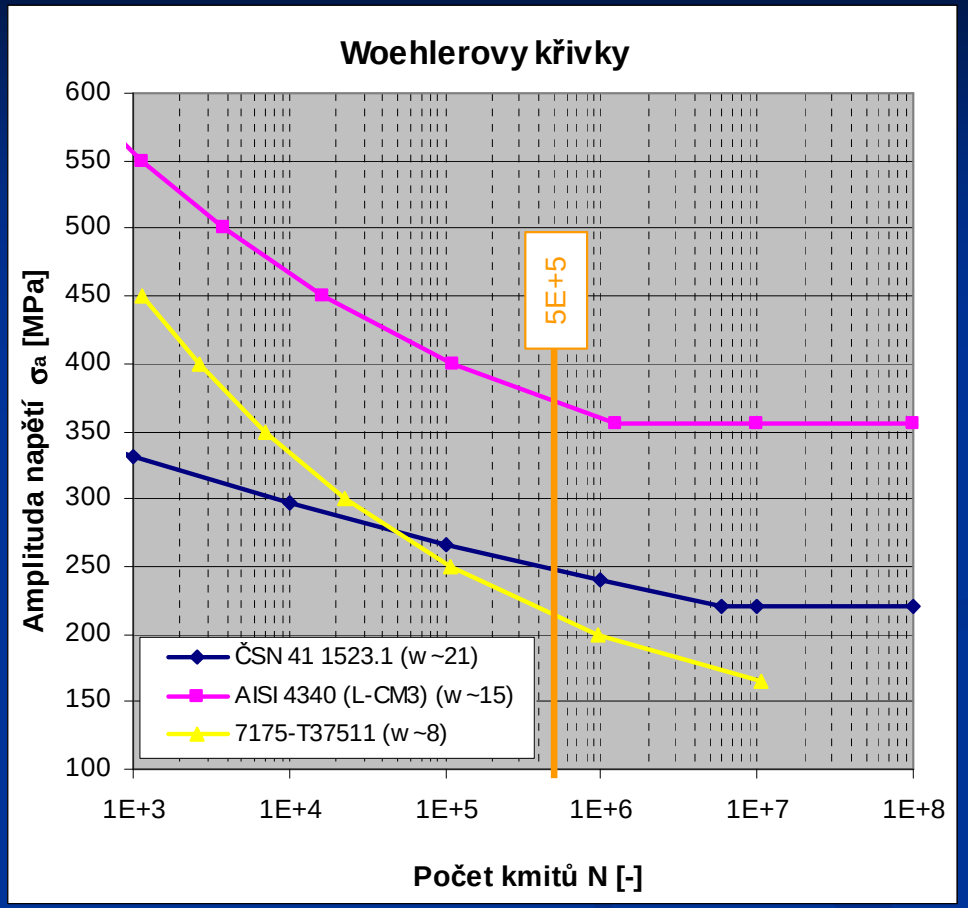
Essentials



- FE-model
  - Topology
  - Results
- Material data
- Knowledge about load history
- Availability of suitable calculation method

FEM-Model Requirements

- Adequately detailed mesh around the critical localities
- Use the submodel of the critical place if available
- Some rough rules:
  - A quarter of circle ~ 5 elements
  - Avoid linear tetrahedrons
  - At least three elements over the thickness, otherwise shells



Load increase	Decrease of fatigue life		
	ČSN 41 1523.1	AISI 4340	7175-T37511
0%	100%	100%	100%
2%	66%	67%	82%
5%	36%	38%	62%
10%	13%	17%	39%

Use of Result Data – Part I

- Results can be printout at
  - nodes of elements
    - can be significantly discontinuous (coarse meshes)
    - the measure of conservativeness can be doubtful
  - integration points
    - the only "exact" values
  - nodes as averaged value
    - smooth in comparison to nodes of elements
    - the same results on edges
  - element centroids
    - used e.g. in FemFat on shell elements and welds (the load values on edges are not decisive – structural stresses)

- more points to be analyzed
- much slower calculation

Use of Result Data – Part II

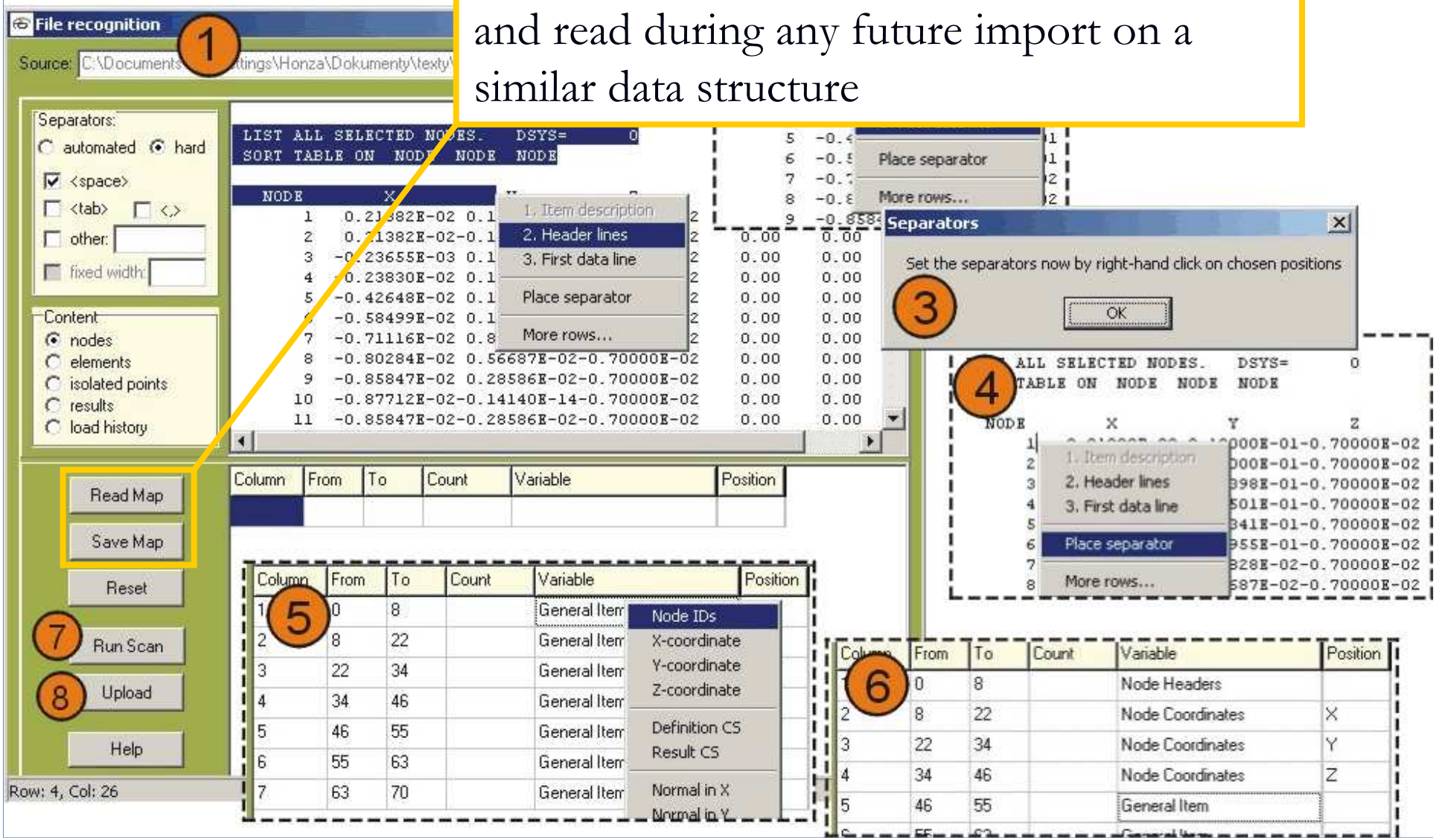
- Shell elements
  - obvious demand by the most of industrial partners (automotive, airplane, transport, etc. industry)
  - further complication as regards listing – results on top and bottom sides
  - questionable results at typical crack initiation areas – junctions, toes, etc.
    - these localities are moreover often affected by welds
    - use of results at element centroids should be preferred
- Bar elements?
  - does the computation make sense?
  - description of potential notches on elements close to zero (if not input e.g. in the fatigue post-processor)
  - notches at junctions are not described in acceptable detail

Data Import to PragTic

- FE-data
  - commonly formatted (rows & columns) ASCII file (solid elements only)
  - MSC.Nastran (*.nas/*.dat files - topology, *.pch – FE-results)
  - ABAQUS (ASCII file *.fil) – suspended at present (is there anybody to sponsor it?)
- Isolated points
  - direct input through the PragTic's interface expected
  - possibility to import as a commonly formatted ASCII file
- Other entities
  - load regimes, calculation methods – possibility to read data from other PragTic's tasks

Example of ANSYS Input

Operations done in points 1-6 can be saved and read during any future import on a similar data structure



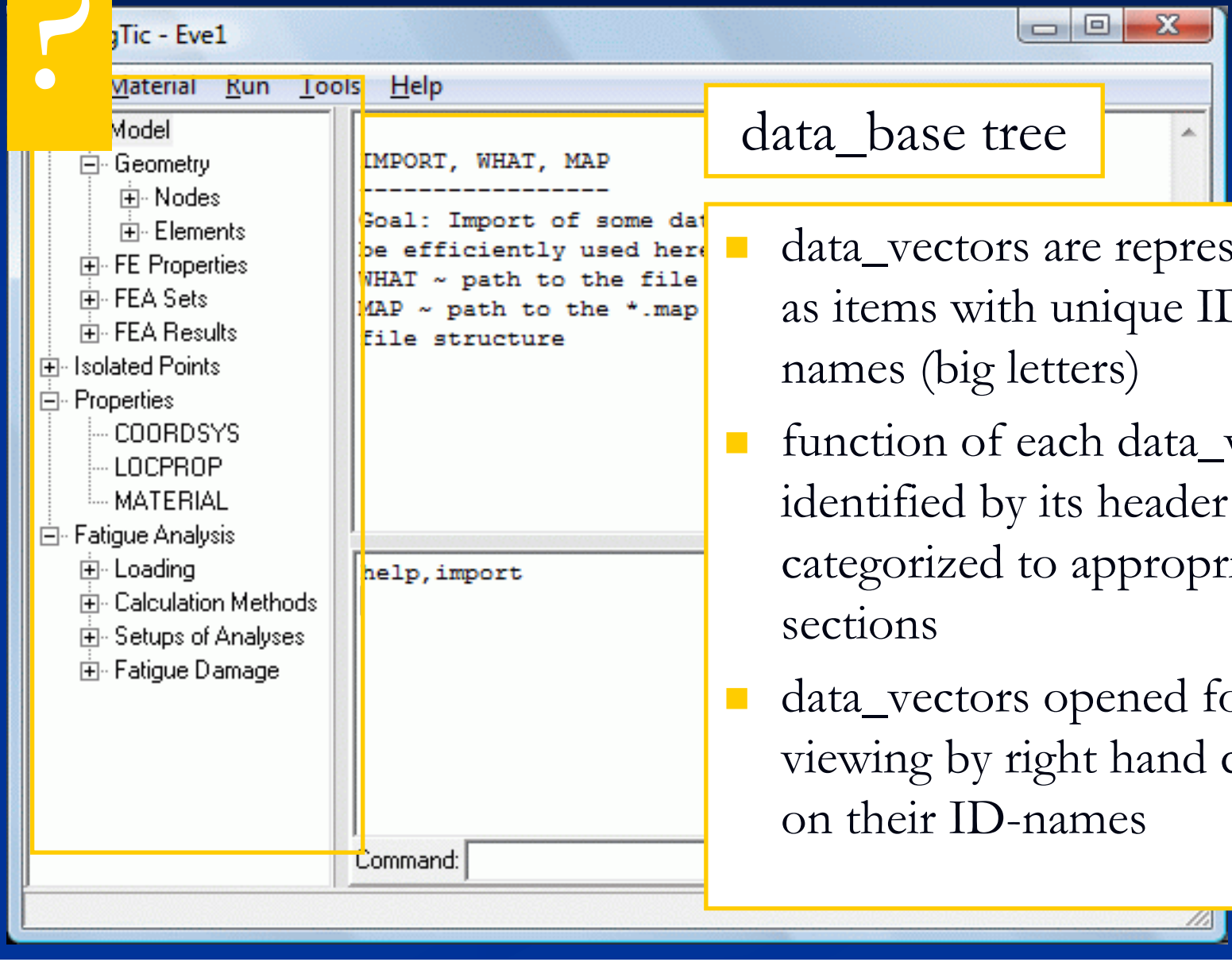
FE-Model Description

- Topology data
  - Node description (ID, coordinates)
  - Element description
    - ID
    - Related material
    - Related element group
    - Related real constants
    - Related coordinate systems (definition, output)
    - Related nodes (element table of incidencies)
- Result data

Data Structure of Tasks in PragTic

- hard-disc representation
  - *.fdb file with description of the task (data_base)
  - directory * (the same name as the task) full with binary files of individual items in the data_base (data_vectors)
  - each file consists of data of uniform type and length
- memory representation
  - data_base – specialized data_vector describing complete content of the task
  - data_vector
    - class build around the data read from files
    - data are cached, read only their part, not the whole file
- PragTic interface representation

Interface



data_base tree

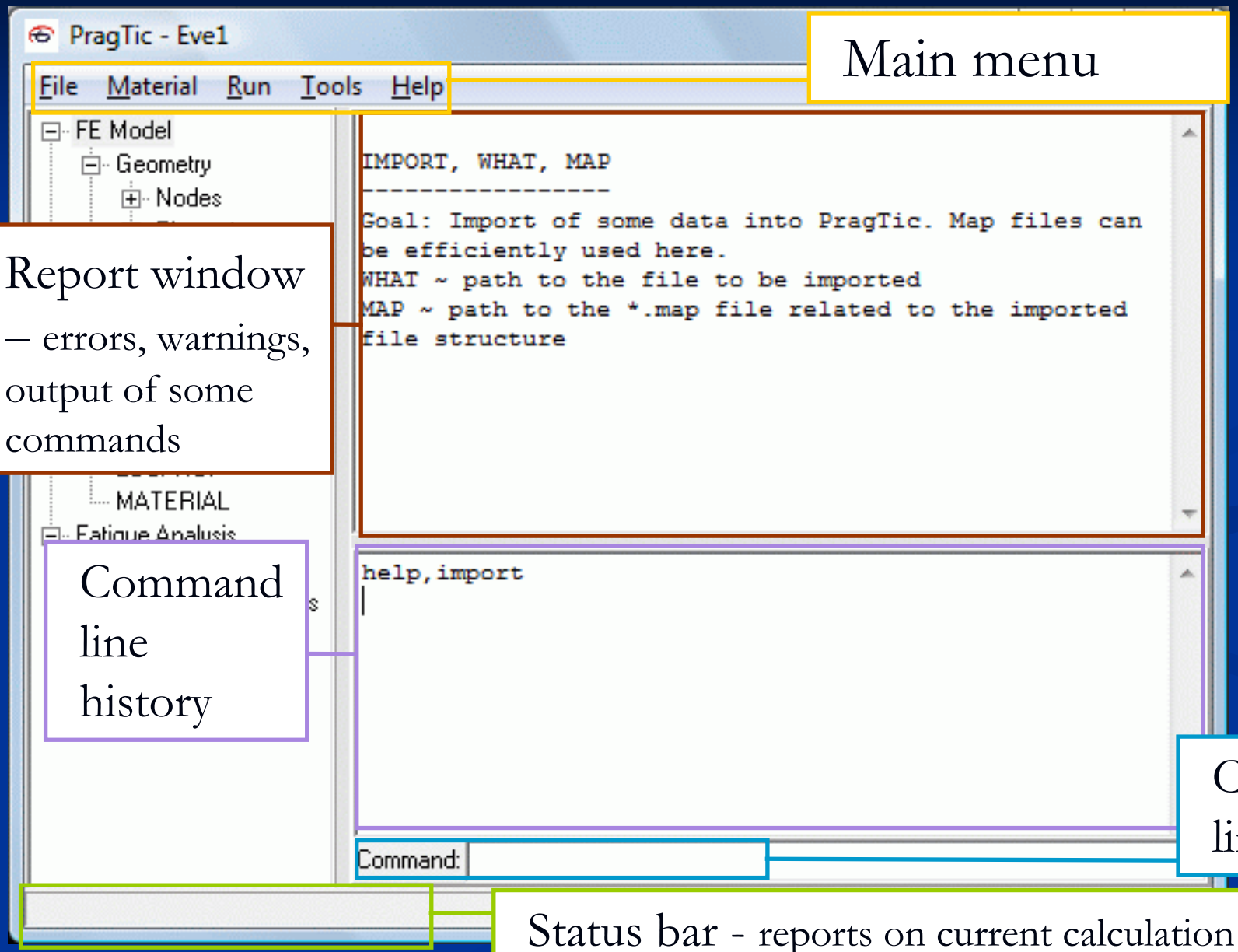
- data_vectors are represented as items with unique ID-names (big letters)
- function of each data_vector identified by its header and categorized to appropriate sections
- data_vectors opened for viewing by right hand click on their ID-names

Interface II – Popup Menu

When complete:

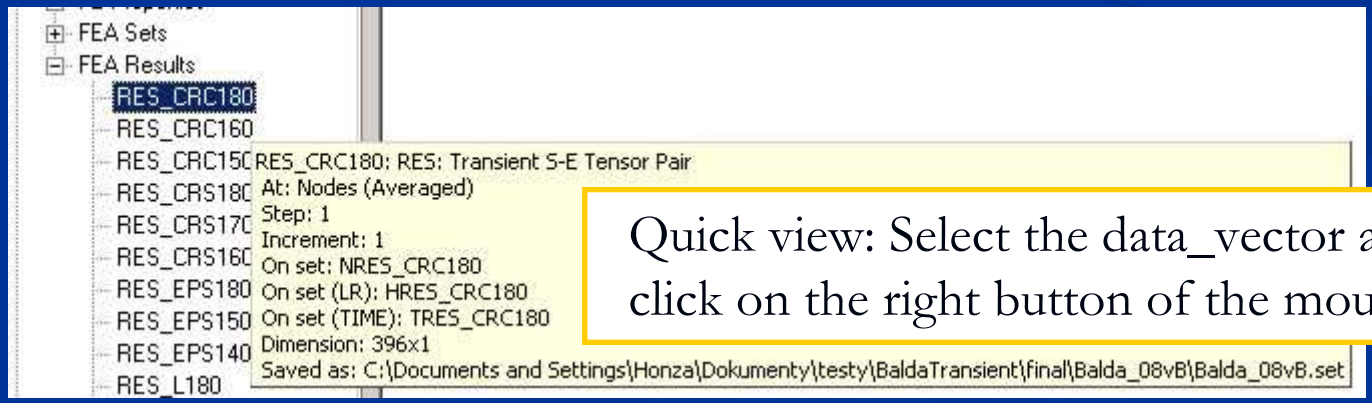
- Opened by a click on the right mouse button
- Differs for various selected items
- **Remove** command
  - removes the data_vector from the task, but leaves the file representation on the hard-disc
- **Delete** command
  - removes also the file
- **Note:** Some data_vector serve as a synergic part of another data_vector – you will be warned that the delete/removal is forbidden

Interface III



Data Items Differentiation

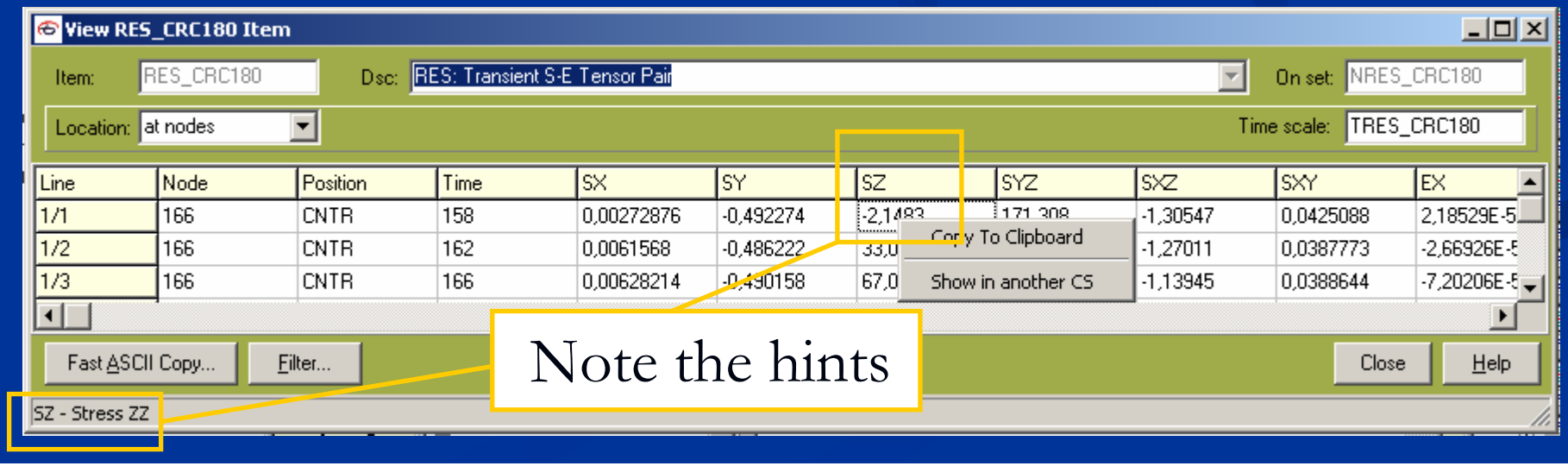
- Each data_vector has a header at its beginning, where are among others data on:
  - Its meaning (e.g. Stress, Strain, Nodes, etc.)
  - Related location (e.g. variable described at element centroids)
  - Number of items, dimension of one item
  - Related sets (e.g. nodal)



Quick view: Select the data_vector and then click on the right button of the mouse

View / Edit Window – Type I

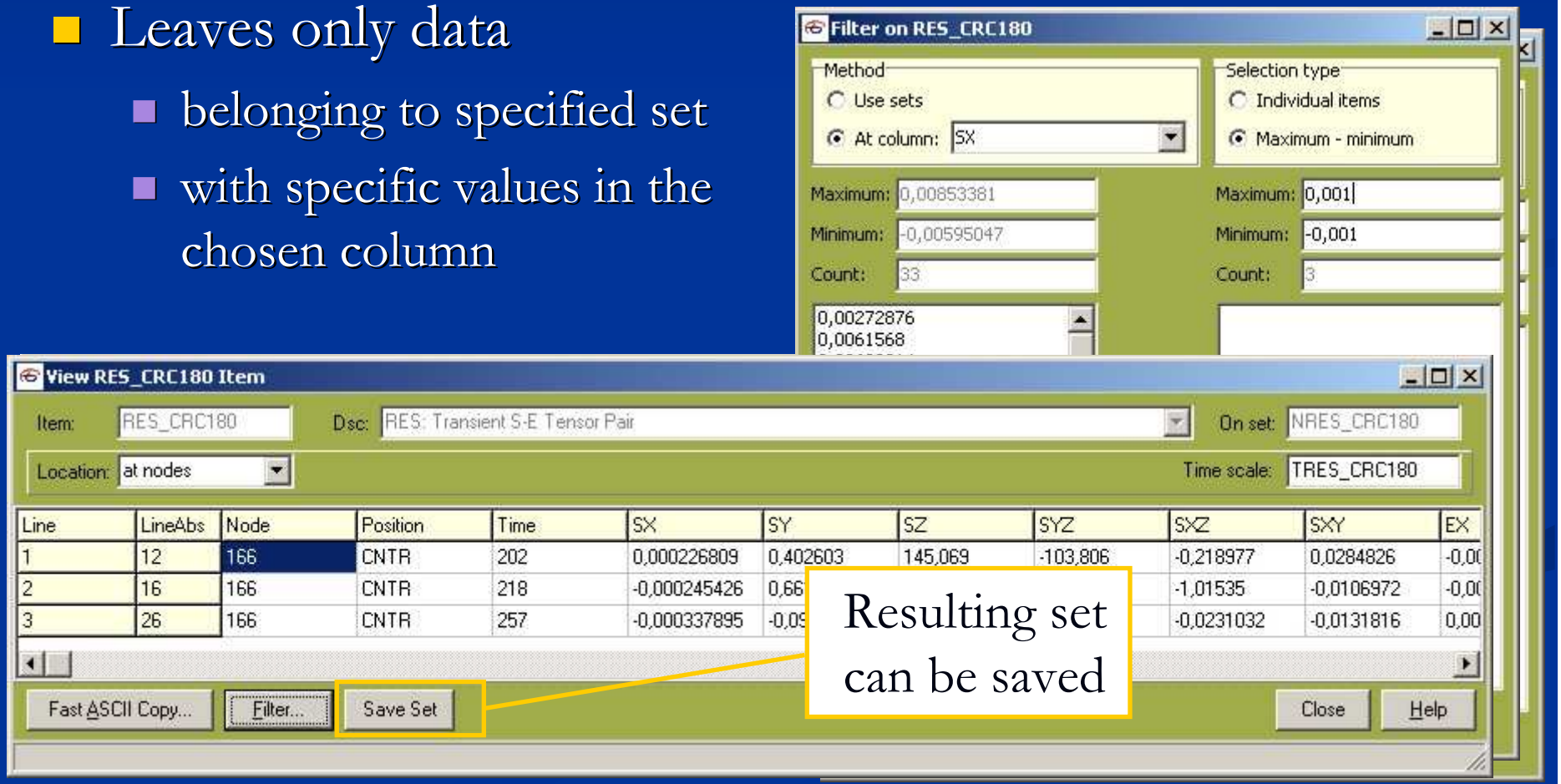
- Description of elements, nodes, isolated points, sets, results, loads
- Use of clipboard enabled
  - Ctrl+C (or the command from the right-hand click menu) for View mode
  - Ctrl+V (or the command from the right-hand click menu) for Edit mode
- **Fast ASCII Copy** – copies whole content of the table to a file, using the tabulators and enters as separators
- **Filter** – see its strength on the next slide



View / Edit Window – Type I

Filter

- Filtering can be used in a consecutive series
- Leaves only data
  - belonging to specified set
  - with specific values in the chosen column

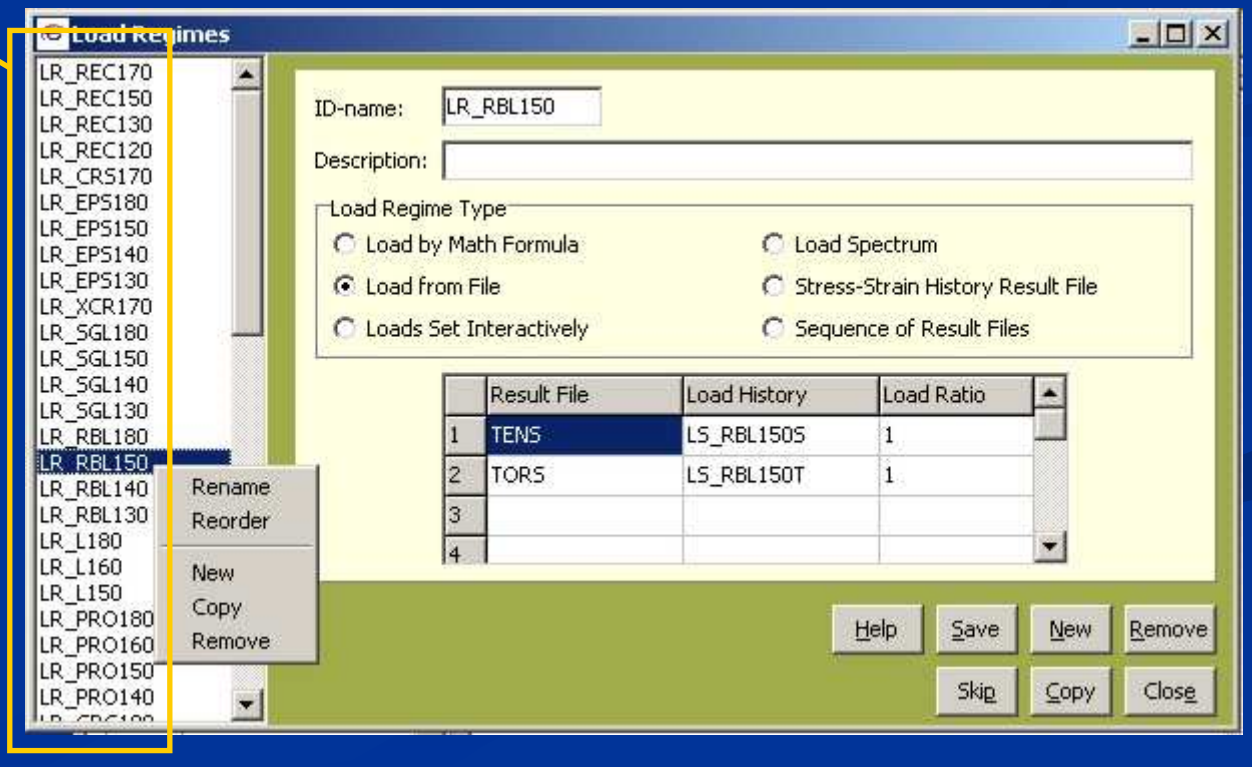


View / Edit Window – Type II

- no command *Back* or *Undo*, every *Save* is final
- not *Saved* changes can be returned back by the *Skip*
- Changing the ID-name to a new one and save is not equal to rename but to a copy to a new item

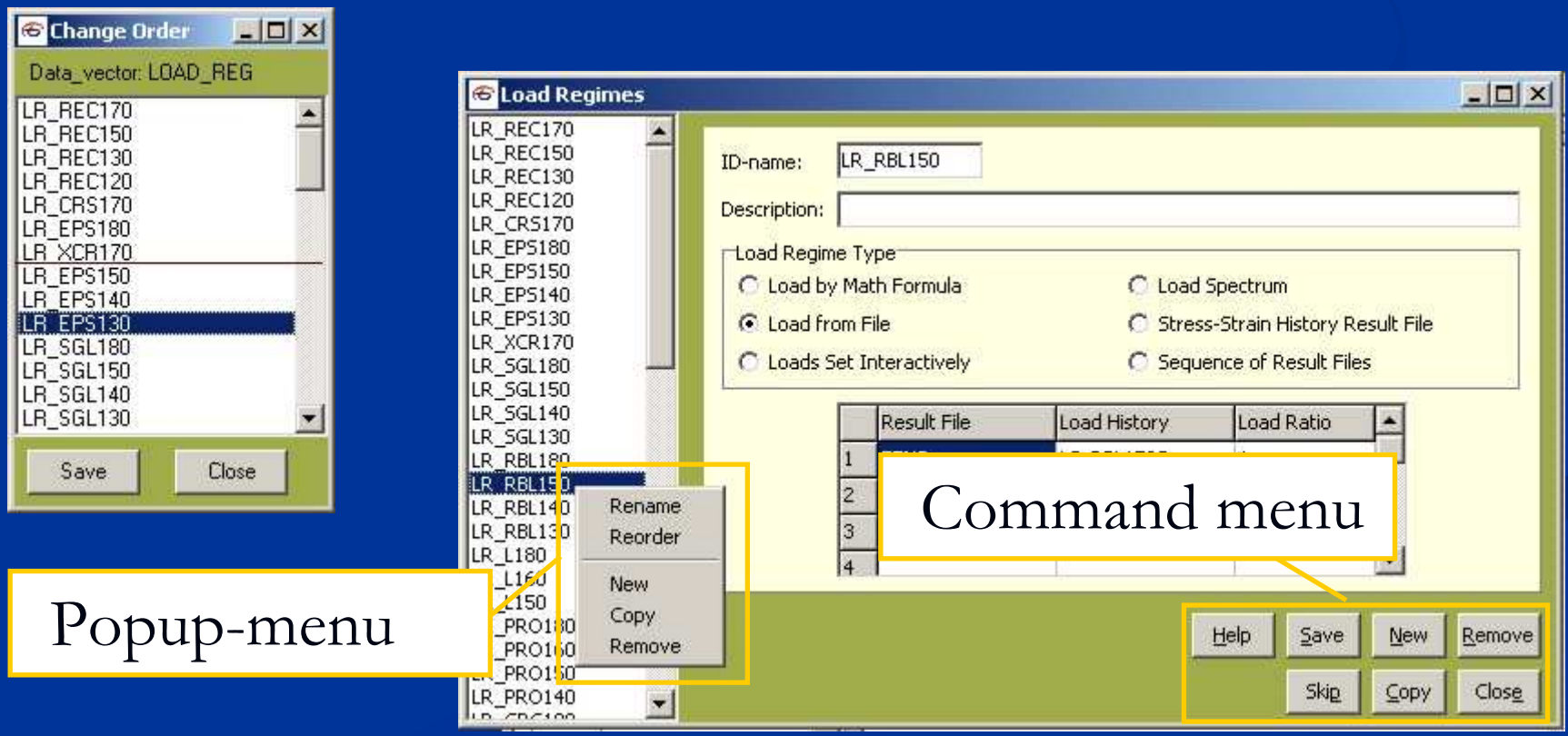
individual items

edition of each item in the data_vector (load regimes here) has to be finished by *Save* if you want to continue to another one



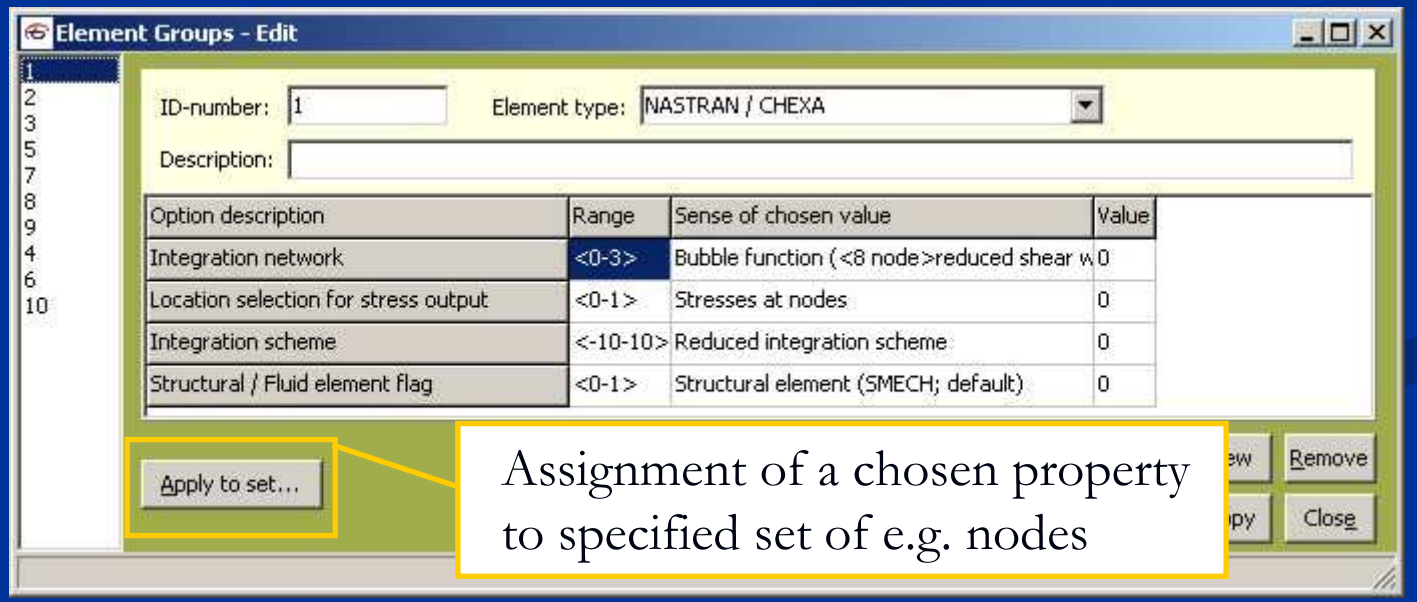
View / Edit Window – Type II

- Popup-menu enabled only for Edit mode
- Some items in the Command menu enabled also in View mode
- **Reorder:** Currently enabled only for Load Regimes and Setups of Analyses



View / Edit Window – Type II

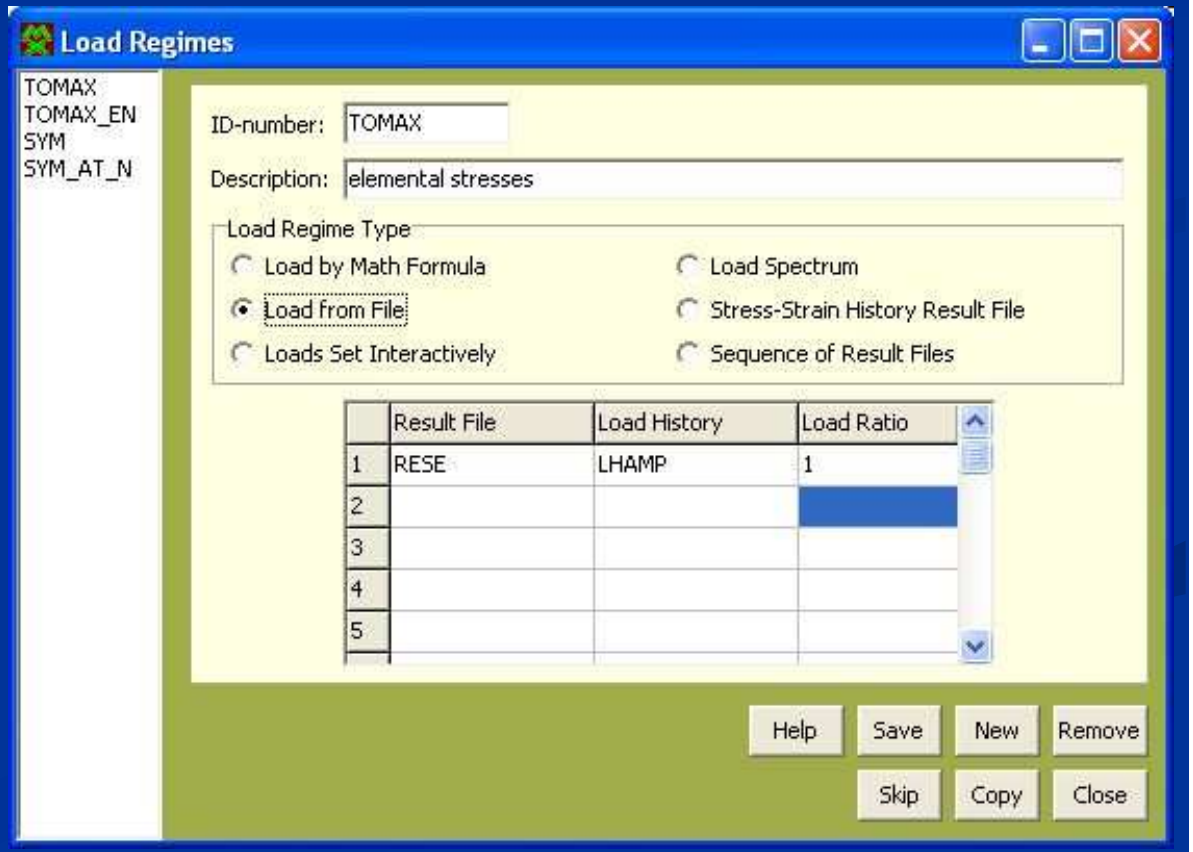
- **Apply to set** button
  - Enables correction of FE-model properties, if they were inadequaely imported to PragTic



Loads Description

- **Load Regime** (LOAD_REG)
  - a combination of all entities necessary for creation of local load histories
    - extern load history + FE-results file + knowledge of load applied to the FE-model
    - local load history in hte FE-result file = transient analysis

- extern loads (Hooke)
  - math formula
  - load by data sequence
  - load spectrum / rain-flow matrix
- transient analysis (no Hooke)
  - local load history read from FE-solver (ABAQUS)
  - **In preparation:** buildup of transient analysis from a sequence of FE-results



Loading

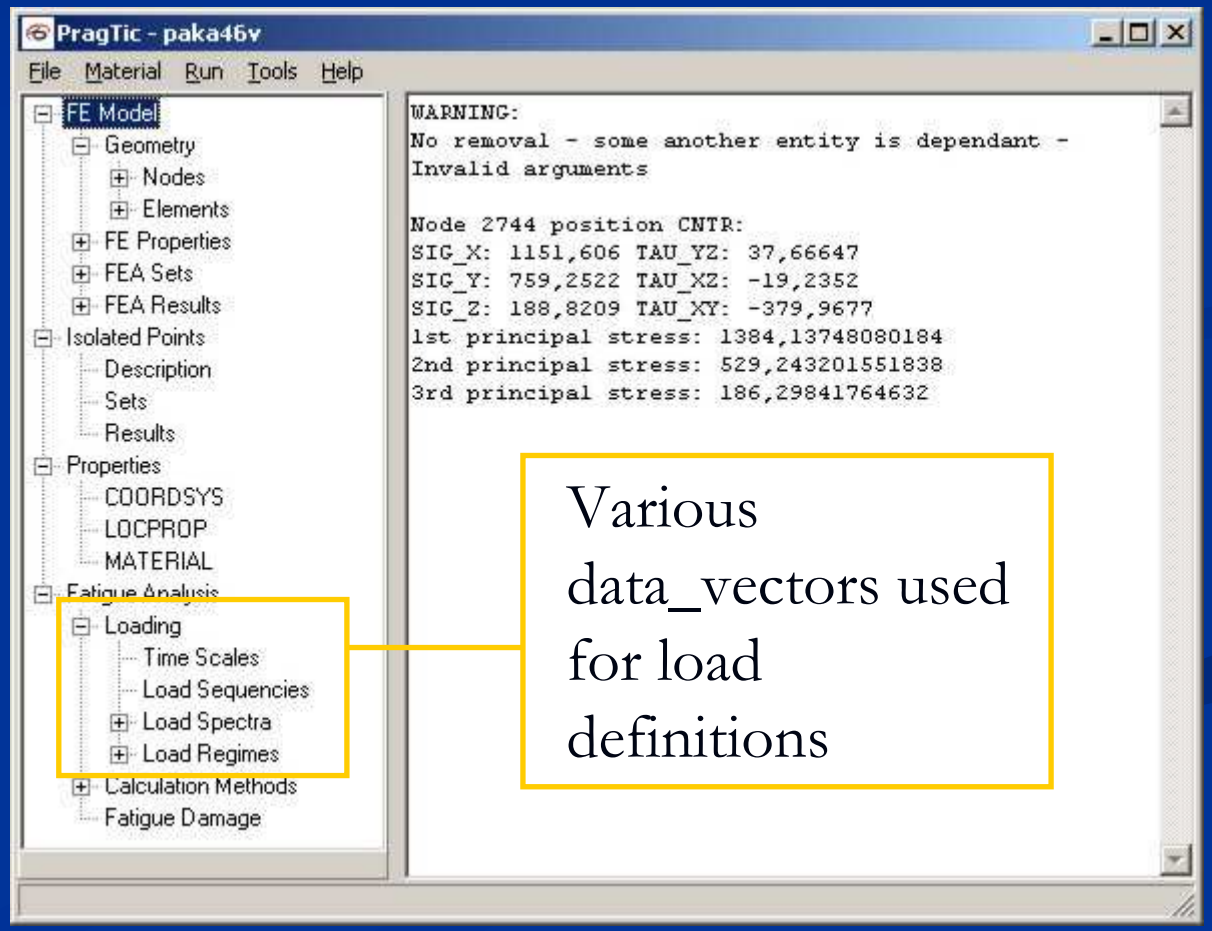
- Linear FEA
  - The individual load channels can be combined by simple superposition
  - If the resulting loads pass over the yield limit, some kind of reduction to elastic-plastic value can be necessary
- Non-linear FEA
  - No superposition allowed
  - More challenging way due to necessity to include all the interacting channels and contacts into one model

Load History Definition

- Transient analysis – local loads over the structure
- History of the acting load – must be in a known relation to the load acting on the FE-model
  - real record (time – force)
  - mathematical formulation
  - load spectrum (upper force – lower force – occurence)
    - load spectra as e.g. required by standards
    - rain-flow matrices

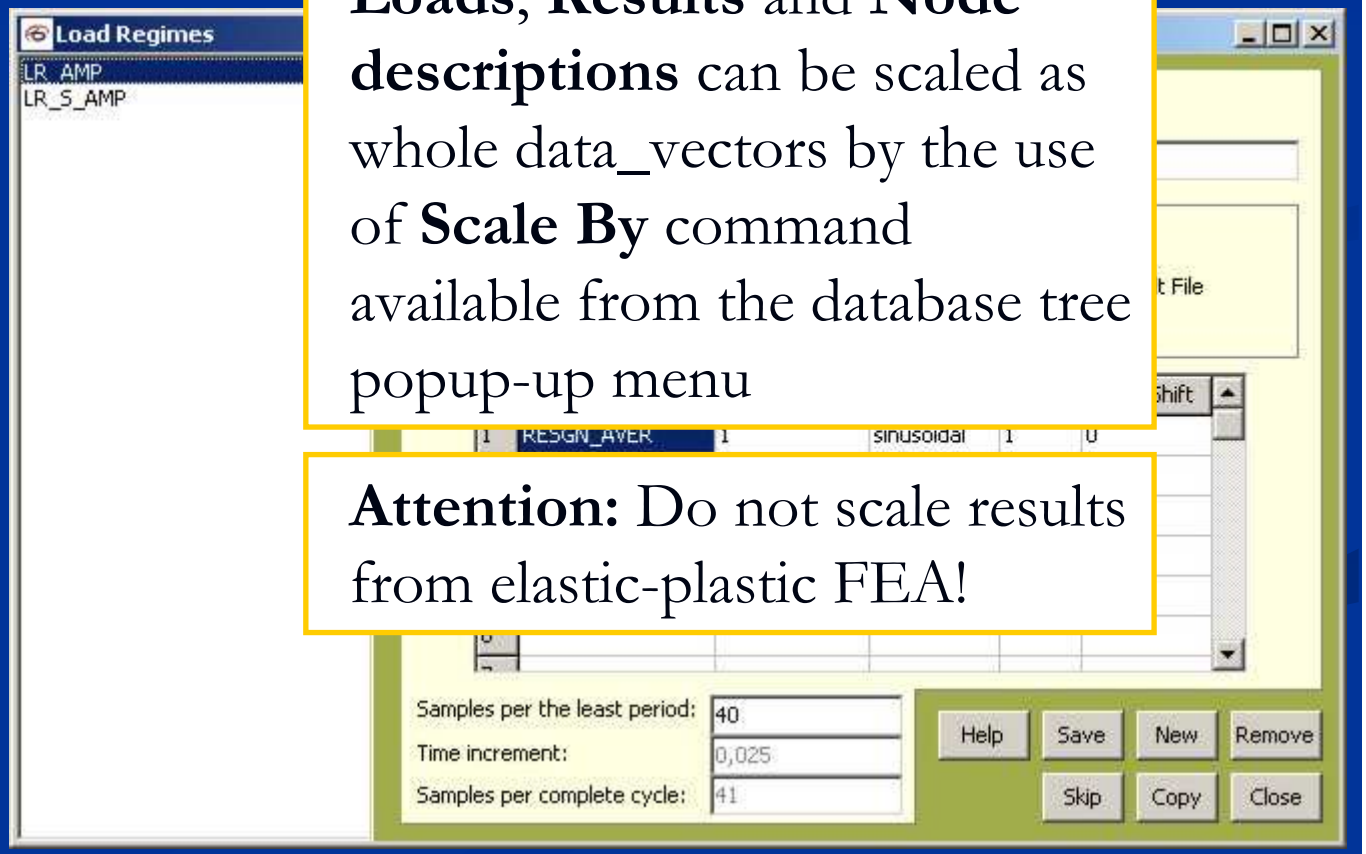
Load Input to PragTic

- Use of the **Import** function from the main menu
- Creation in PragTic (**New** command at the right-hand click menu opened on **Time Scales, Load Sequencies** and **Load Spectra**)
  - direct editing
  - use of Copy and Paste function (Ctrl+V) at the void cells of Edit window – be sure that the size of the clipboard and the edit window coincide



Load Scaling

- Optimum setup:
  - FEA solution for unit load (e.g. 1 N, 1 kN, etc. – something with what can be easily worked further)
  - FEA result input for a desired load
    - e.g. the load equal to the load amplitude
    - not so handy for further modifications



Loads, **Results** and **Node descriptions** can be scaled as whole data_vectors by the use of **Scale By** command available from the database tree popup-up menu

Attention: Do not scale results from elastic-plastic FEA!

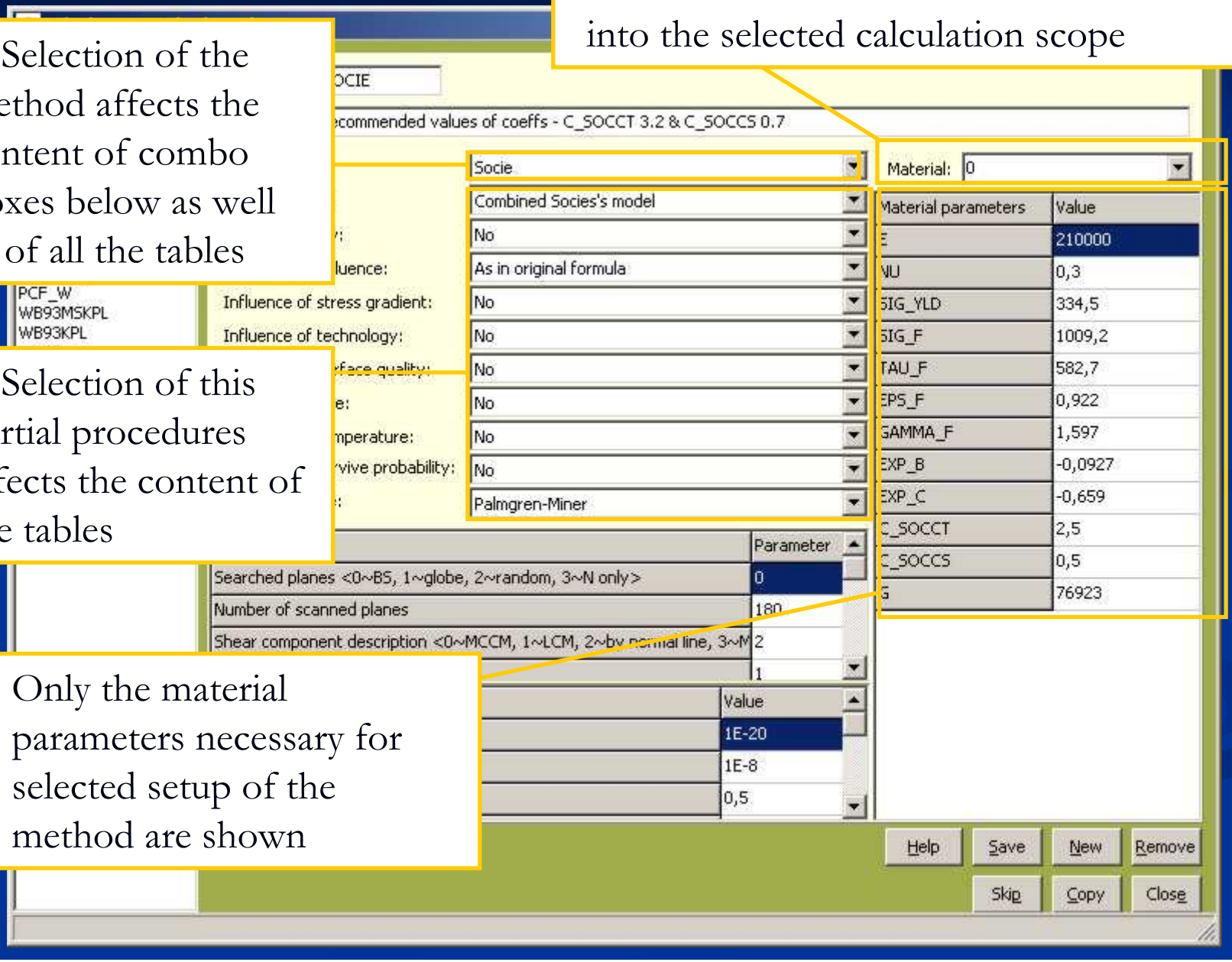
Setup of Calc

Check before any calculation that the M.Ps. are defined for all materials included into the selected calculation scope

1. Selection of the method affects the content of combo boxes below as well as of all the tables

2. Selection of this partial procedures affects the content of the tables

Only the material parameters necessary for selected setup of the method are shown



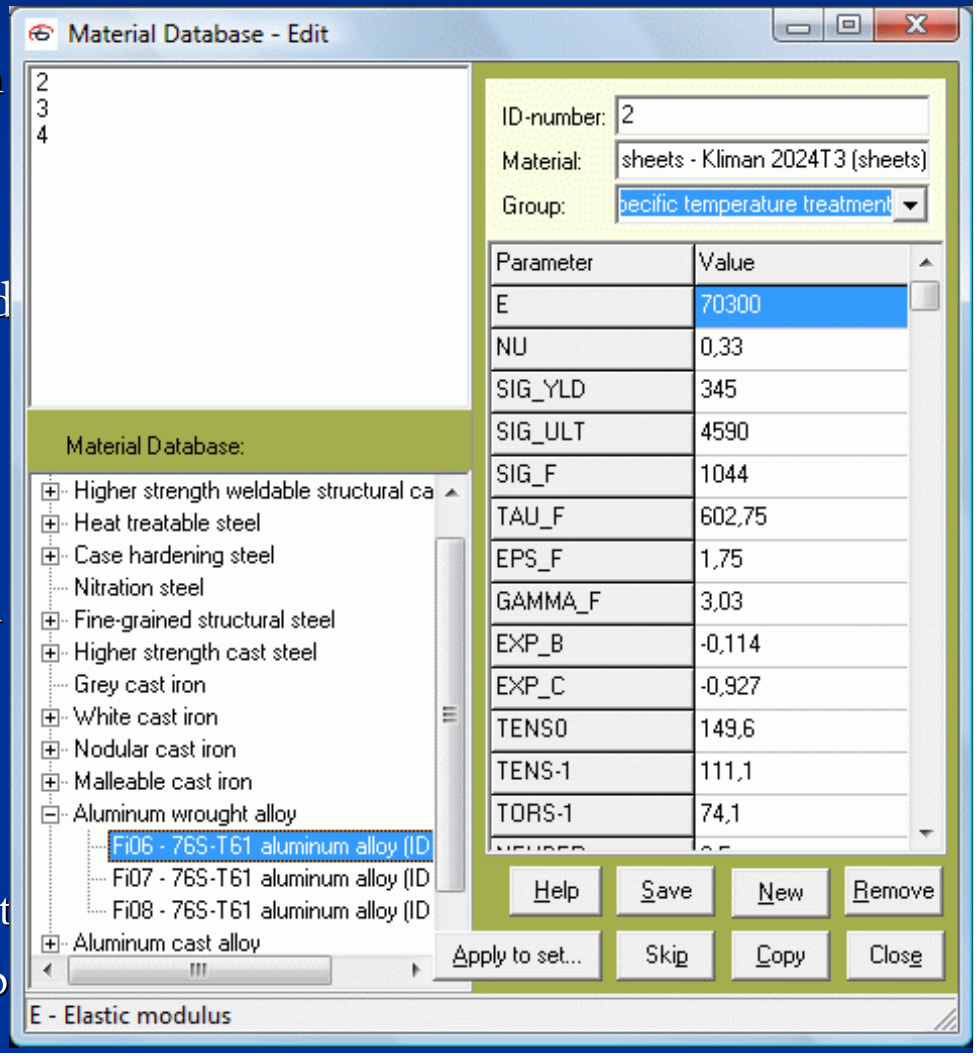
Transient Analysis

- Local load history built for cases, where the linear Hooke's law is not valid
- Two ways of creation
  - local load history recorded for each node/element (FE-postprocessor builds the transient analysis)
  - set of results at different times chained together (fatigue solver builds the transient analysis)

Transient S-E Tensor Pairs (SET_Pair item) are placed to the **Results** / **FEA Results** groups and not to **Loads**

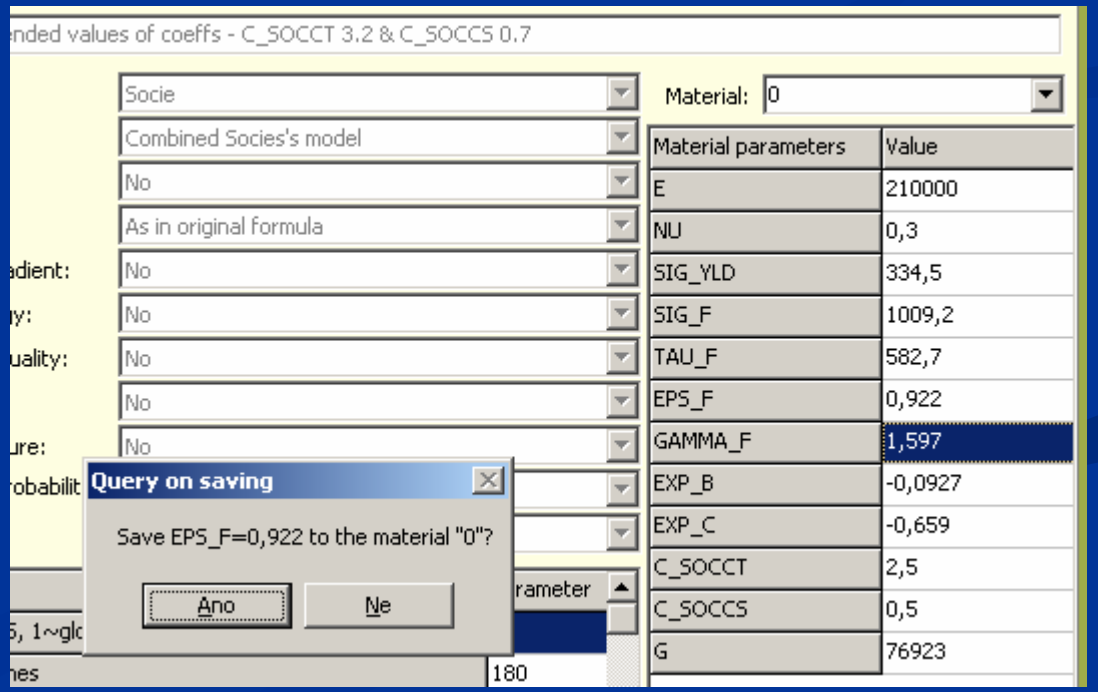
Material Parameters
(MATERIAL)

- Even a complete import of the FE-model e.g. from *.nas file provides only several static parameters
- Material definition common also with Isolated Points – it should precede before their definition
- Material parameters definition opened from:
  1. **MATERIAL** ID-name of data_vector
  2. **Material** item of the main menu
- Some material parameters preset in dependency on the chosen material group
- **Note:** Use of a decimal comma or dot depends on your local Windows setup



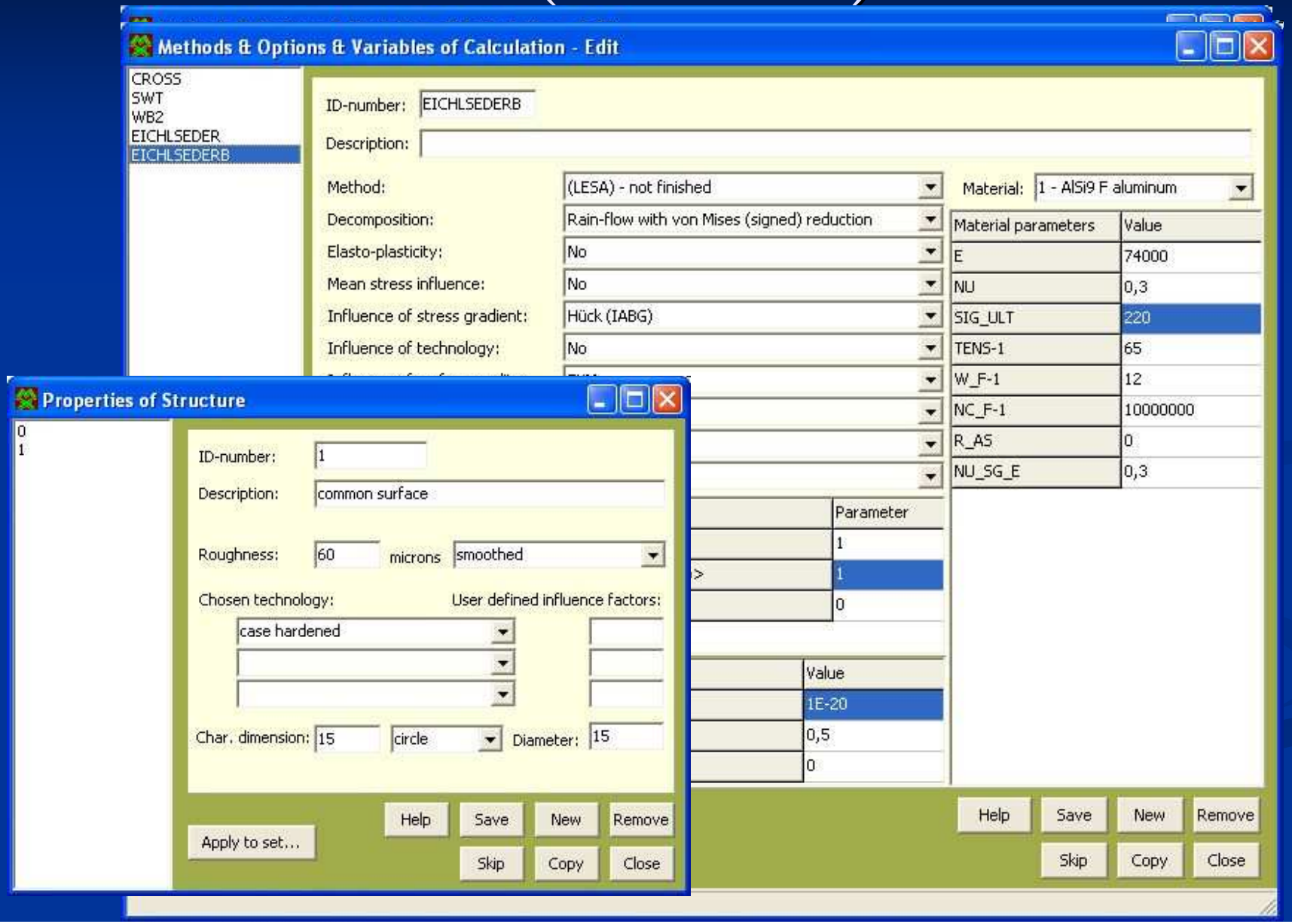
Eventual Setup of Material Parameters

- Use the Methods dialogue
- You are not defining the setup of methods, i.e. Methods can be opened even in the otherwise passive View mode
- You can see much more efficiently, which material parameters you really need in order to get to results
- Any your change of a table cell starts a dialogue asking whether to save your input into the Material data_vector



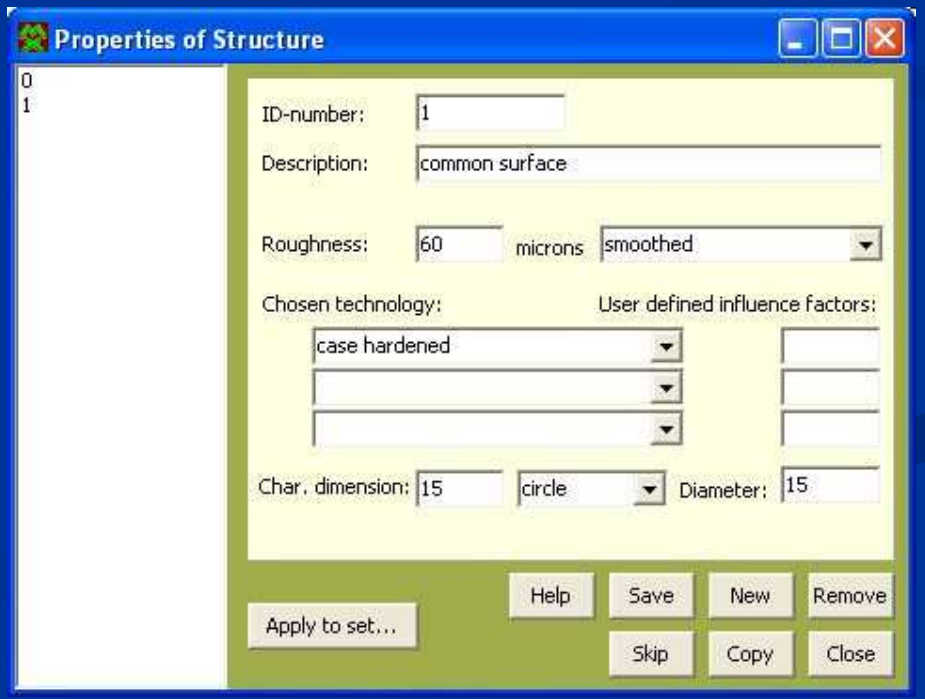
Setup of Calculation Methods

(METHODS)



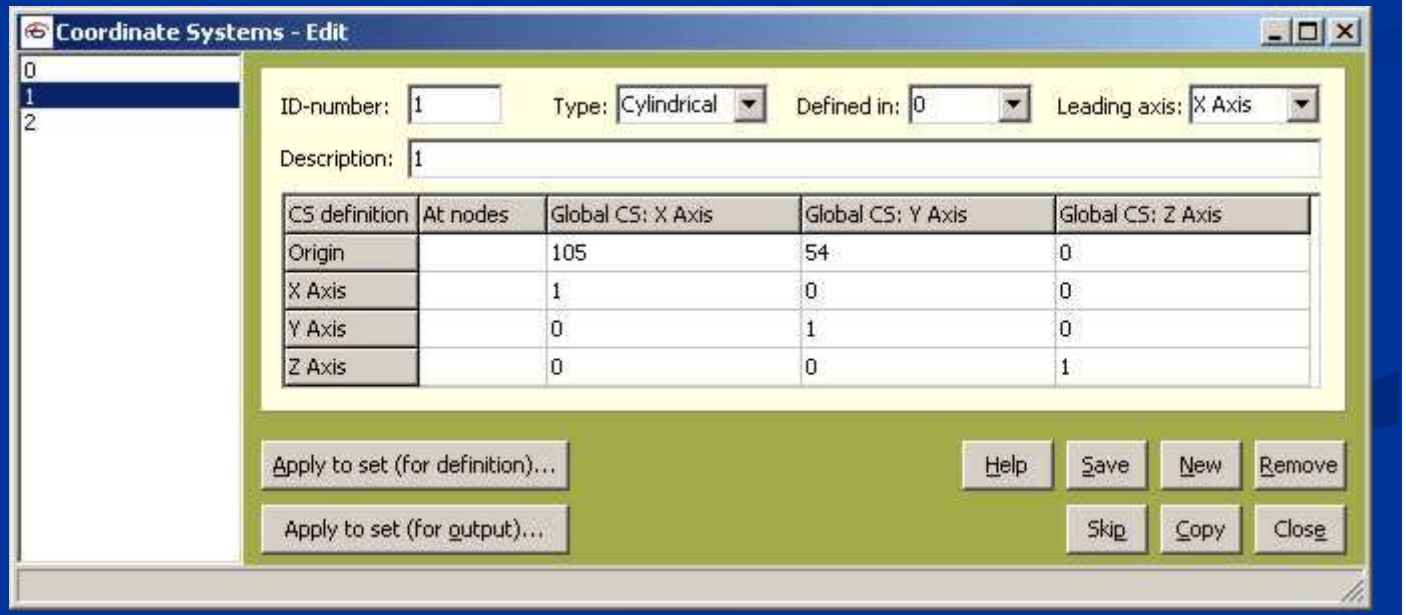
Properties of Structure
(LOCPROP)

- The dialogue covers facts on the specimens that are included neither in the FE-model nor in material parameters description
  - Surface quality
  - Up to 3 technologies affecting the surface layer
  - Size effect
- Values in text fields override the selected items in comboboxes
- Activated for LESA method only



Coordinate Systems
(COORDSYS)

- New coordinate systems can be defined on basis of other previously defined coordinate systems
- Used both for Isolated Points and the FE-model
- Definition of a new item:
  - the values in the first column (nodes, IPs) override the next positions
  - 3 rows only define the c.s.

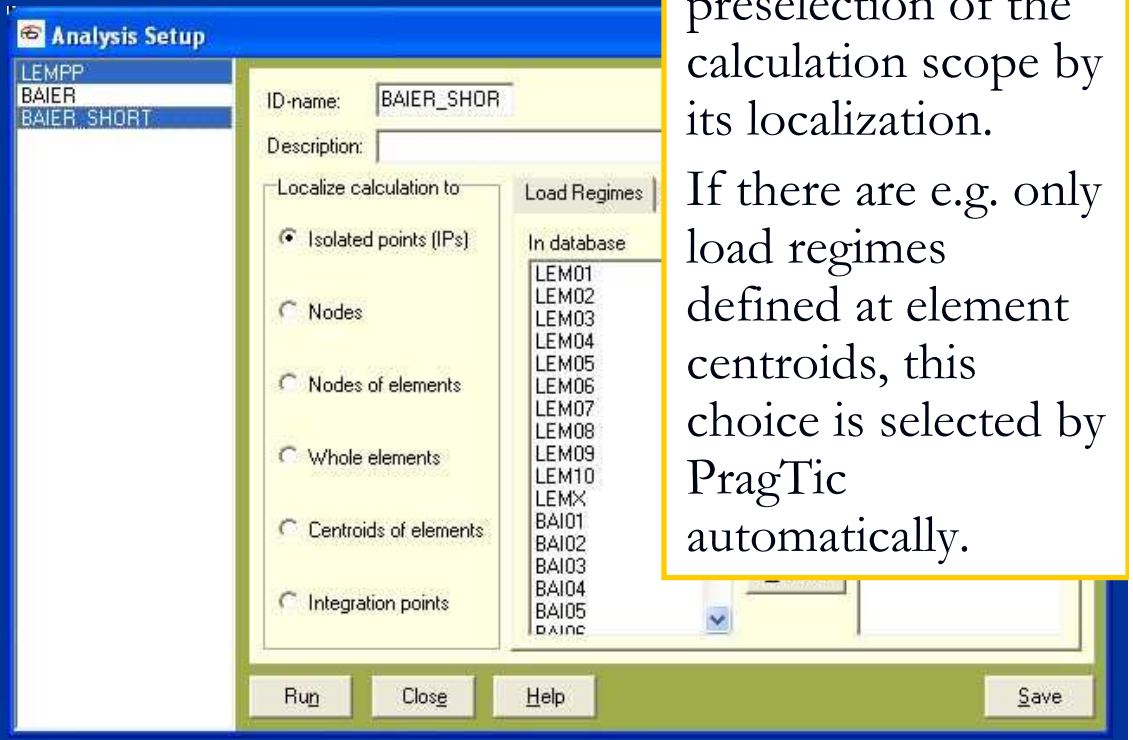


Coordinate Systems
(COORDSYS)

- Thanks to **Apply to set** buttons the c.ss. can be applied to imported data additionally
- **Results**, **Loads**, **Node descriptions** can be transformed in the View mode to another c.ss.
- Unsolved yet:
  - The results imported to PragTic could be printout in another c.s.
    - A further data_vector descriptor marking the related c.s. has to be implemented in some next PragTic version
  - The results on shell elements can be defined in their native coordinate systems (i.e. cs intrinsic to every particular element). Note that
    - it should not affect the fatigue calculations
    - but it affects transformations between individual c.ss.

Setups of Analyses
(ANA_SETUP)

- Specifies the focus of the calculation:
  - where (localities)
  - on which data (load regimes)
  - by which method (methods)
- Each setup consists of:
  - at least one item from load regimes
  - at least one item from methods
  - at least one set of nodes, elements or isolated points
- Calculations loops
  - for each load regime
    - at each point
      - for each method

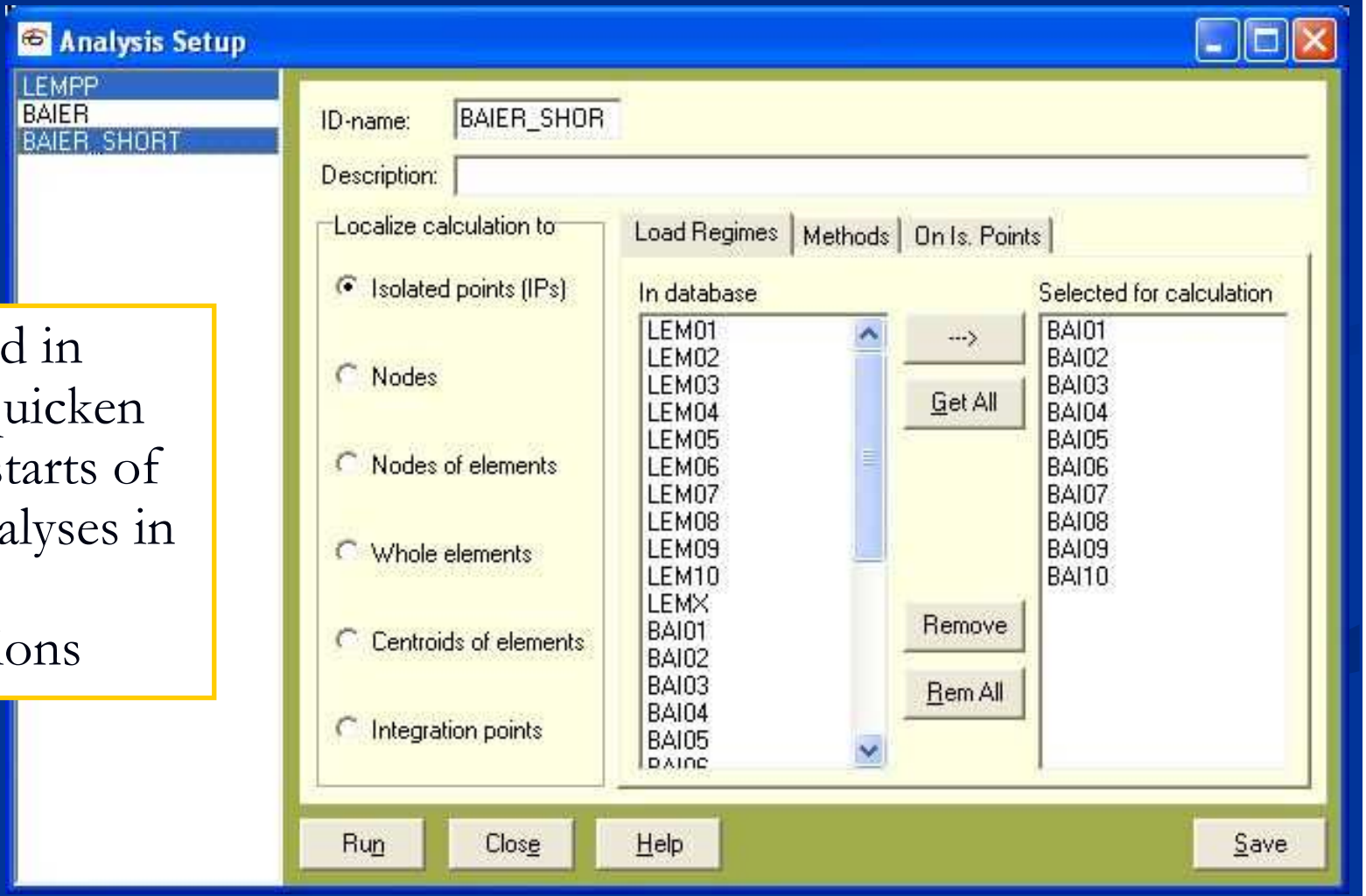


Note the preselection of the calculation scope by its localization.

If there are e.g. only load regimes defined at element centroids, this choice is selected by PragTic automatically.

Setups of Analyses
(ANA_SETUP)

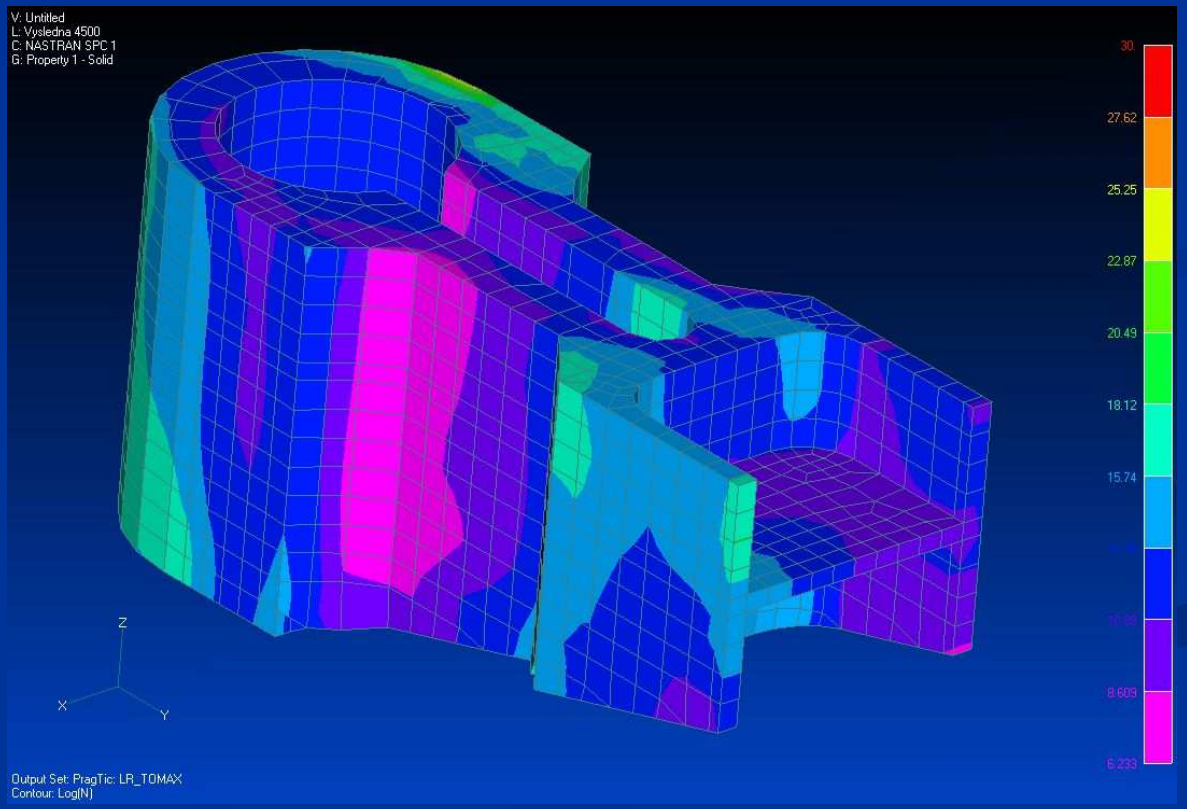
- Replaces the common **Run** window in PragTic v.0.2betaH



Introduced in order to quicken repeated starts of fatigue analyses in various modifications

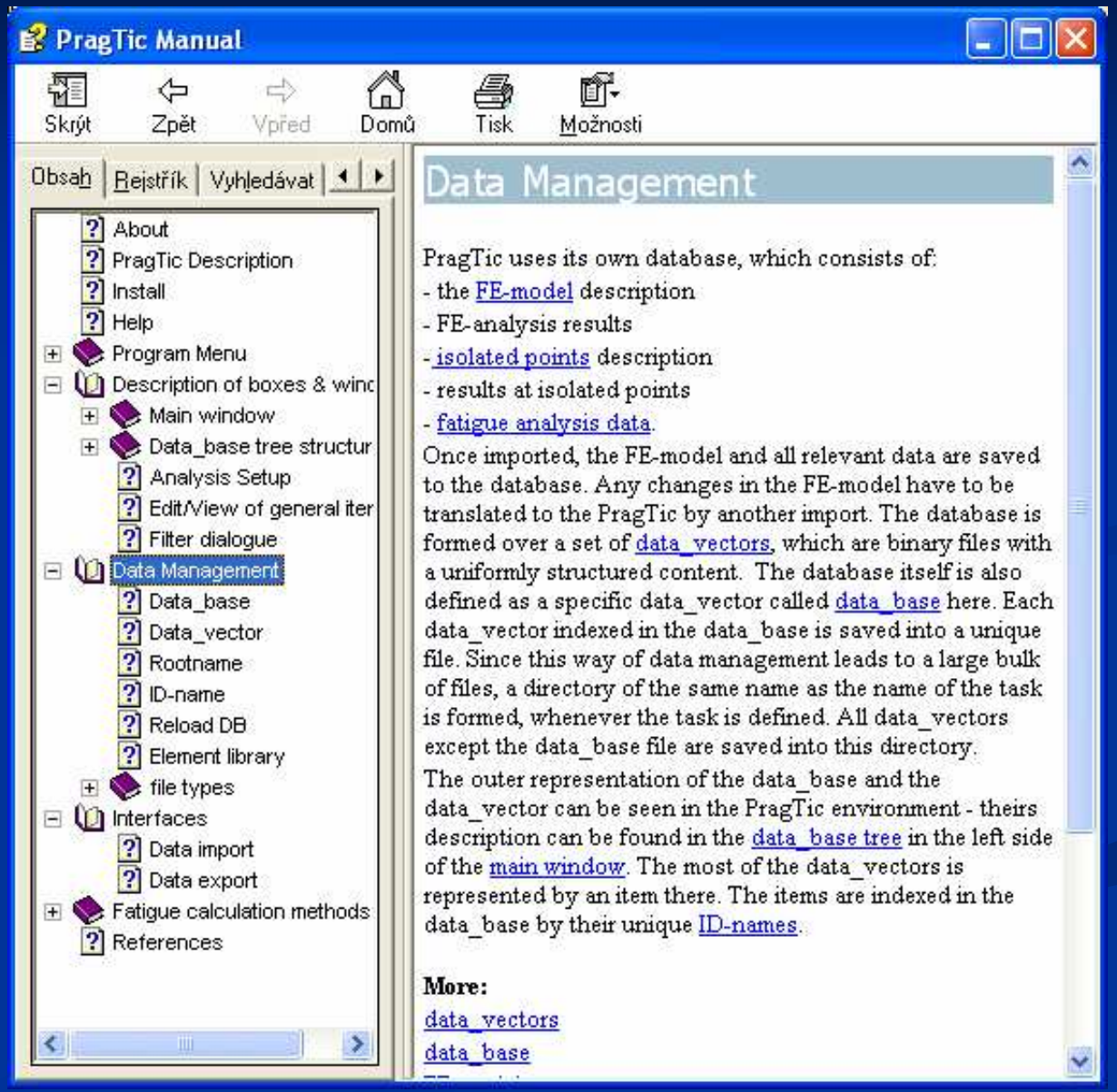
Results of Fatigue Calculation

- Spreadsheet summary of damage and calculation related variables
- Graphical interpretation of results:
  - import back to the original FE-post-processor
  - export of fatigue results into a FEMAP neutral file (*.neu)
  - export to a batch file that can be imported to Ansys



PragTic's Help

- freely downloadable
- created as a context help
- *.chm version
  - common context manual in Windows
  - started from the PragTic's interface (Help buttons?, Alt+H, F1 key)
  - HELP command at the command line
- *.html version
  - either for download or accessed via Internet (www.pragtic.com)
  - content coincident to the *.chm version
  - does not have Index, thus any search without knowing the structure is problematic



New Features from the Last Meeting

- Rain-flow
  - Accessible in the Tools section to be run without real fatigue calculation
  - RF matrix can be shown
- Material Database
- Command Line