

QUIKFRAME xp is a program for the linear elastic and 2nd order analysis of 2D structural frames in any material. Its novel approach means you just sketch your frame, sketch your loading and view the results. No need to mess around with co-ordinates, you don't even need node or member numbers. Just create a few construction lines and snap members to them. Powerful 'drag n' drop' copying and mirroring facilities help you create and load complex frames in no time at all. Sections may be chosen from the built-in libraries, imported from the companion MERLIN/quikEC3 programs or input from the keyboard. No need to remember complex sign conventions, just point and click to attach loads in the correct orientation.

An analysis is performed automatically whenever you want to view results and, uniquely for a low-cost program, **QUIKFRAME xp** has several 2nd order analysis methods. Using the latest recommended techniques, these are vitally important for today's slender and lightweight structures.

QUIKFRAME xp has an all new interface using the latest software techniques. Optional 'plug-in' modules for steel design (BS5950 and EC3) and timber design (BS5268) are fully integrated, offering the complete analysis and design package

- **Fast and friendly**
Create frames in a fraction of the time using **QUIKFRAME xp's** intuitive 'sketch and click' approach.
- **Surprisingly powerful**
All the geometry and loading facilities you need without any built-in limits
- **Includes 2nd Order Analysis**
Absolutely essential for today's lightweight long span frames
- **Integrated Design modules**
Optional fully-featured steel and timber design requires very little additional input
- **Very Clear Output**
Practical, easy-to-follow results presentation with output to CAD packages

To evaluate **QUIKFRAME xp**, please visit our website at www.gtscad.com or call the number below for a free CD. The evaluation software is time and usage limited but is otherwise fully functional. Take a look at the other programs in the **QUIKSOFT** library: **QUIKPORT** - the ultimate portal program, **QUIKJOINT** - the most comprehensive connections program, **QUIKBEAM** - the project-based steel beam program and **MERLIN/quikEC3** - the electronic blue book + custom sections + simple calculations.

Full Technical Details overleaf

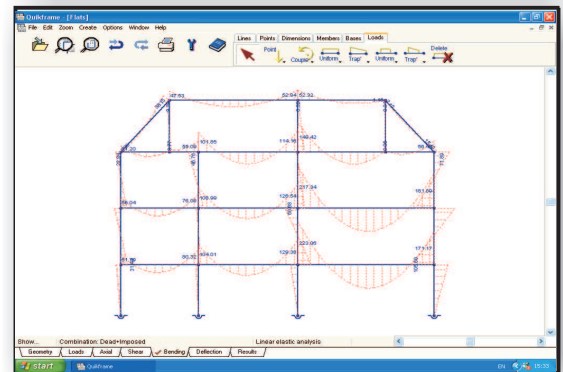
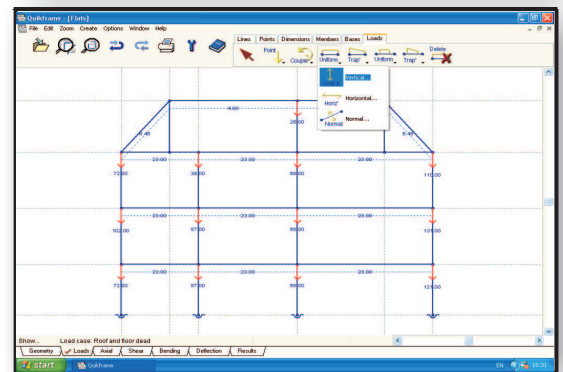
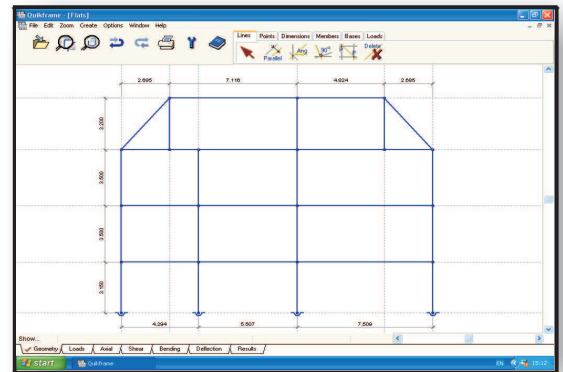
GTS CADBUILD Limited

Woodbrook House, 30 Bridge Street, Loughborough, Leics LE11 1NH

Tel: 08456 250559 Fax: 01509 269221 Email: sales@gtscad.com



QUIK frame xp



Supported Operating Systems

Windows 2000, XP and VISTA. *Virtually any computer capable of running these systems will be satisfactory. QUIKFRAME xp is designed for standalone or network use.*

Types of Frame

2D with uniform or tapered straight members. Member ends may be fixed or pinned. Bases may be pinned, fixed, sliding or spring.

Units

kN, m, cm, mm.

Materials

Any elastic material may be used and section tables for steel and timber are included. The steel tables include UK and European sections with a limited selection of USA and Japanese sections. Custom sections may be created in the companion MERLIN/quickEC3 programs and imported.

Project Files

QUIKFRAME xp is project-based and shares the project folders of other QUIKSOFT software to permit data interchange. Multiple calculation sheet headers can be entered which is useful for the free-lance Engineer working for several clients.

No limits

Unlike earlier versions, **QUIKFRAME xp** has no built-in limits - its scaleable architecture automatically adjusts to suit any size of problem.

Drawing Methods

Orthogonal and inclined construction lines, points. Click on points or intersections to create members. Bases and loads are then added with a few mouse clicks. Geometry and loading may be moved, copied, rotated and mirrored.

Loading Types

Uniform, trapezoidal, patch and concentrated applied vertically, horizontally, axially and normally to a member. Moment about a point.

Analysis Model

Unlike conventional programs, **QUIKFRAME xp** creates its own analysis model, saving the Engineer considerable time. It is not necessary to worry about co-ordinates, node or member numbering and members don't have to intersect at member ends. The drawing model is first checked for errors and then converted to an analysis model. In preparing the model for analysis, the program automatically splits members with intermediate connections. Loads are also split and re-assigned. Tapered sections are modelled as short elements each with the appropriate properties. This gives more accurate analysis than many programs without any additional input from the user. Joints are created and automatic bandwidth optimisation carried out. All results are presented for the drawing model so the user doesn't really have to know anything about the complexities of the analysis.

The Analysis Engine

Analysis is carried out by a state-of-the-art analysis engine incorporating various second-order methods in addition to the traditional linear-elastic methods. By using the appropriate method(s), the Engineer can have more confidence that his/her structure will meet all stability requirements. This is vitally important for today's slender lightweight structures and mandatory for some.

Linear-Elastic Analysis

This is very fast and is accurate where overall or in-member deflections are small.

P-Delta Analysis

An elastic analysis is performed to predict deflections. The predicted deflections are then used to modify the analysis model which is re-analysed and a new set of predicted deflections calculated. This is repeated until the largest difference between the assumed and predicted deflection is insignificant.

No account is taken of in-member deflection and hence induced additional moment due to the axial load (p-delta)

p-delta analysis

This method is very similar to the P-Delta method except that the in-member effects are accounted for by splitting members into shorter elements. It is a good general purpose method for frames with large deflections but can be slow owing to the considerable amount of extra computation involved.

Stability Analysis

QUIKFRAME xp's stability analysis uses the Livesley and Chandler stability functions. These stability functions account for the loss of stiffness in members due to increased axial load commonly referred to as *S and C* functions.

An analysis using stability functions requires a good estimate of axial load before the analysis is performed. This is achieved by feeding in the axial loads from a previous analysis and repeating until an insignificant difference exists between estimated and calculated axial forces. To put it another way: *at each load level the analysis is repeated until a consistent set of forces is obtained.*

This is a fast method which accounts for elastic instability and P-Delta but does not cater for in-member effects.

Elastic critical load factor analysis

The aim of this analysis is to determine the lowest factor (*Lambda*) for the applied loadings such that the frame is just stable.

The applied load factor (*Lambda*) is incremented by a small amount and the stability analyses repeated until a **consistent** set of forces is obtained, or until the stiffness matrix yields a non positive determinant indicating elastic instability.

Once elastic instability is detected the program repeats with values of *Lambda* between the last stable and unstable analysis. The program stops when the changes in *lambda* are insignificant.

Steel Design (optional)

QUIKFRAME xp's built-in steel designer requires very little input. Just modify any of the default restraint conditions as necessary and use either the DESIGN or CHECK wizard. The comprehensive and neatly presented results will satisfy any checking authority. It's also very easy to try alternatives in order to achieve the most rational or economic solution. BS5950 or EC3 versions available.

Timber Design (optional)

This fully integrated module requires little additional input and will carry out a full design or check existing timber sizes in accordance with BS5268. A wide range of timber types and sections are accommodated. Output is very clear and similar to manual calculations, ensuring ease of checking. Uniquely, it will work alongside Steel Designer to handle frames with a mixture of steel and timber members. Load factors are automatically adjusted accordingly.

Results

Unlike other programs, **QUIKFRAME xp** doesn't have a separate analysis phase, it analyses when you ask to view results. In this way, you can always be sure that the results are for the currently displayed frame no matter how many changes you may have made. Most frames will analyse in a flash but some forms of second-order analysis will take longer. As **QUIKFRAME xp** checks your frames before elastic analysis, you will get no annoying analysis failures with cryptic error messages. If, during second order analysis, your frame suffers elastic instability, analysis will stop with a clear warning.

When you are happy with the results from your frame, a full or selective set of calculations may be viewed on screen or printed. The screen image will (within the limitations of your system) be exactly as it will appear in print. The frame may be exported to CAD using DXF and the member forces may be passed to QUIKJOINT for connection design.

Why buy from GTS?

GTS has supported structural, civil and geotechnical software for twenty-five years. Its efficient support service is highly regarded by its large customer base in over 60 countries worldwide. No call centres, no waiting on hold, no auto-replies to your emails - just fast, friendly assistance from qualified and experienced Engineers.

All purchasers automatically become members of the GTS Software Club with the following benefits:-

- Free program updates and enhancements
- Free telephone, fax and email support
- Free insurance against software loss (dongles excluded)
- Special Offers on new software and other selected items.

The free subscription period is usually 12 months but may be shorter on special offer software. Subscriptions may be renewed at very reasonable cost.