PRODUCT MODEL FOR STRUCTURAL ANALYSIS OF STEEL FRAMES

Markku Heinisuo

Juha Hyvärinen

Mika Salonen

Rakenteiden Mekaniikka, Vol. 32
Nrot 1-2, 1999, s. 4-24

ABSTRACT

The paper introduces a product model for transferring data from CAD-applications to FEM-packages. The product data model is presented using EXPRESS (and EXPRESS-G) data definition language (ISO 10303) and the corresponding data is stored using the standard file format ISO-10303-21 (known as STEP-files). The product model is a part of a Knowledge Based approach to the design of steel structures. The pilot programs are written to import the corresponding STEP-file of FST-model (product model including the geometry of the skeleton, exported from CAD) into KB-system, and to export the input file for analyzing programs. The present paper motivates the use of entities needed in the structural analysis of a steel skeleton. Explicit stiffness matrices (based on the solutions of the governing differential equations) are recommended to be used whenever they are available in order to keep the model as small as possible. Similar product models (for geometry and analysis) with slight modifications can be used for other skeletal structures, wood, concrete, composite, etc.

INTRODUCTION

Many solutions in closed systems for the data transfer between different design tasks e.g. geometrical modeling and structural analysis, have been made for design of timber [1] and steel structures [2]. The "closed system" refers to an interface between a specific CAD software and a specific finite element program. Another way to solve the problem under consideration is integration of analysis concepts within a CAD software [3, 4]. An integrated architecture for the structural analysis of mechanical structures has been proposed [5]. The structural analysis can be done (following the previous reference) not only using FEM but also using strength of material approach or boundary element method or something else. There exists a novel knowledge-based assistance for finite element modeling [6], which can be used not for the selection but for the *construction* of the analysis model from the geometrical model. Intermediate files are used for communication between the finite element program and the modeling framework. A solution for the data transfer problem in an open (i.e. hardware and software independent) environment nowadays is using neutral data files.

Many neutral files are used for the data transfer within different software environments [7]. Product models can also serve as specifications for such files. In this paper a product model mainly for structural analysis of steel frames is proposed. There exist at least two models for structural steel frames where the geometry and the analysis of steel skeleton have been considered [8, 9] and where the standard ISO-10303 has been applied. ISO-10303-11 [10] and ISO-10303-21 [11] have proven efficient and widely accepted "tools" to present product models and they are used in this paper also. The analysis models given in the references [8, 9] are based on the use of beam elements. Moreover, the geometrical model and the structural analysis are integrated in those references.

A proposal for the application independent standard of finite element analysis does exist [12]. At the present, that model is perhaps too large to be applied in practical steel design in building and construction projects. The situation is continually changing due to the increasing power of computers. There exist general algorithms for the generation of three dimensional analyzing models from the geometrical model [13], but these algorithms also lead to too heavy models in practice. Some features are taken from the models in the literature into the present model.

OBJECTIVES AND REQUIREMENTS FOR THE PRODUCT MODEL

One part of this research project was to develop a product data model for the analysis of steel skeletons which can be used for data transfer in the knowledge based controlled steel skeleton design process and which covers the most urgent needs in practice. It is believed, that there must be some knowledge based (or similar) tool for the creation of the analysis model from the geometry due to the complexity of geometry. This is true especially for the joints of the steel skeleton. When modeling the joints of a steel skeleton the very profound models may be needed [14] depending on the level of the information required from the behavior of the joint and moreover, from the whole skeleton. Also, the bars between joints must be analyzed using very profound beam elements in some cases [15]. This situation holds especially for the thin walled purlins with open cross sections used widely in steel skeletons.

A knowlegde based approach (KBS) for the design of steel skeletons has been proposed [16]. The KBS is used for the *selection* of the analysis model from the propositions stored in the database. This system tends to have limited coverage [6] but it seems to work well in the problem considered. The KBS is used mainly for the selection of suitable joints of the steel skeleton taking into account the mechanical and economical aspects. Following that the KBS system proposes suitable joint parameters and the proper analysis models. The user can choose the model using the KBS taking into account the need of the accuracy of the result (0D, 1D, 2D, 3D, local or global models), resources of the project (time, software and hardware capacities, know-how etc).

One goal of the present study was also, that the model should be as computer and software independent as possible. The use of the standard ISO 10303 (STEP) was chosen to fulfill this demand because the use of that standard seems to be applied widely in the field of structural engineering, today. In order to keep the model reasonable in size, the integrated resources of ISO-10303 have not been applied, but only the three parts (11, 21, 104) mentioned above have been used. The analysis is based on the FEM.
The following features were also decided to be incorporated into the model because it had been found, in previous projects, that they are essential in order to perform the practical

design:

1. Unique reference to the geometrical model from the analysis model must exist.
2. There must be many analysis models available.
3. Analysis types must include at least linear and buckling analyses.
4. The analysis model must include submodels with their own coordinate systems, 0D, 1D, 2D and 3D elements must be available with reference to the element type (not in detail as in ISO-10303-104/CD).
5. Many types of beam elements must be available due to different needs in practice and due to interfaces between different level local element models [17].
6. Explicit stiffness matrices (based on the solutions of the governing differential equations) are recommended to be used whenever they are available in order to keep the model as small as possible.
7. Element constraints are needed e.g. when analyzing purlins.

Moreover, the model must be as simple as possible and it must be possible to expand the model to the cases excluded so far from the model. The most urgent needs to expand the model are the loading schema and the results schema which are begond the scope of this paper. Also, gap elements are badly needed when modeling the joints. The model is kept linear and the only non-linearity arises when running buckling analyses. Static analysis is supported at the present stage.

PRODUCT DATA MODEL FOR STRUCTURAL ANALYSIS OF THE STEEL SKELETON

A partial EXPRESS-G diagram of the proposed product data model for the structural analysis of steel frames is presented in Fig. 1, showing the main entities and their principal relationships.

The entities used for capturing analysis model information are:

- STRUCTURAL_ANALYSIS, collecting general information of steel frame and different ANALYSIS_CASEs.
- ANALYSIS_CASE, for each loading case combinations.
- ANALYSIS_MODEL, including description of analysis model, it also informs what coordinate system is used and number of nodes and elements.
- ANALYSIS_TYP, description type of the structural analysis, e.g. is analysis static or buckling.

The EXPRESS definitions of the entities are presented and explained in the following. More details of the model can be found in [18].

```
ENTITY STRUCTURAL_ANALYSIS;
       structural_analysis_id : identifier;
       structural_analysis_assumptions : OPTIONAL description;
       project_id : STRING;
       skeleton_id : STRING;
       skeleton_version : STRING;
       skeleton_version_date : STRING;
       analyses_to_be_run : SET [1:?] OF ANALYSIS_CASE;
  UNIQUE
       UR1 : structural_analysis_id;
END_ENTITY;
```

The identifiers are always structured as alpha-numeric expressions (STRING), which are unique for the current instances of the objects in the scope of the model. The descriptions (STRING) will also include the information of assumptions made for the analysis. The attributes project_id, skeleton_id, skeleton_version and skeleton_version_date refer to the geometrical model. The format to refer to the entities in the geometrical model is to give a unique identification of the objects. Here the skeleton_id will refer to an instance of entity called SKELETON in the product model (FST-model, [16, 19]).

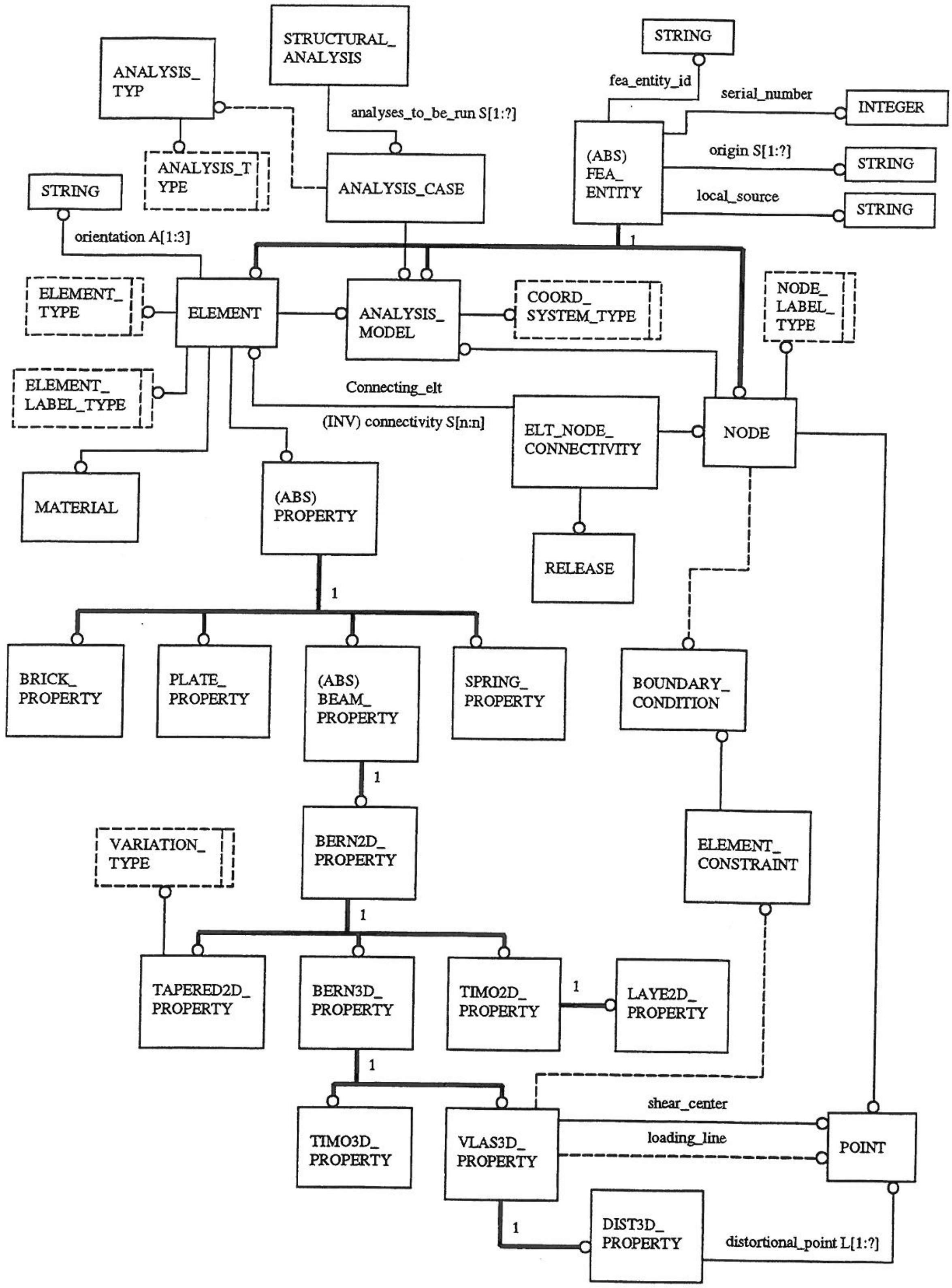

Fig. 1. Partial product data model of structural analysis of skeleton.

```
ENTITY ANALYSIS_CASE;
      analysis_case_id : identifier;
      analysis_case_assumptions : OPTIONAL description;
      serial_number_of_case : INTEGER;
      analysis_model_in_case : ANALYSIS_MODEL;
      analysis_type_in_case : OPTIONAL ANALYSIS_TYP;
      serial_number_of_combin_of_loading_case : OPTIONAL INTEGER;
  UNIQUE
      UR1 : analysis_case_id;
END_ENTITY;
```

If the analysis_type_in_case is not given then only the analysis model is produced (perhaps without loading information). The same ANALYSIS_MODEL and ANALYSIS_TYP may belong to many ANALYSIS_CASEs. The combinations of ANALYSIS_MODEL and ANALYSIS_TYP must be controlled by the KBS. The default value in the programs of the present study is that all the combinations are to be run. Many ANALYSIS_MODEL entities can be generated from the same geometrical model (SKELETON). These partial models can be truss, stiffening plane, one column etc. It must be noted that the same object (in the geometrical model) can belong to many ANALYSIS_MODELS (see the attribute origin of the entity FEA_ENTITY). Also, many ANALYSIS_MODEL entities can be generated from the same partial model of the skeleton. A typical case is the generation of the plane frame and the space frame model (or some other 3D-model) from the same geometry. The KBS must controll the generated analysis model.

Although the loading schema is omitted in the present study, it is necessary to refer to it a little. The loading is divided into two main entities: loading cases and loading combinations. The loading case means the basic loading and its characteristic value without load factors (e.g. self weight, snow, wind etc). The loading here is understood as the loading of the skeleton, not necessarilly the actual load of the structure. There are at least two ways to handle the loading cases: to use the principle of superposition or not. It is possible to apply this principle if the case is both materially and geometrically linear. Some softwares have possibilities to use the superposition. The basic loading cases and the loading combinations are solved by superimposing the basic solutions without solving the total finite element model when combining the load cases. This is a very efficient way to solve the mechanical problems of the skeleton (and of other structures, also), especially when the loading must be considered as a moving load (e.g. live load). Unfortunately, all the softwares do not have this possibility and then the loading cases must be solved separately. When generating the structural analysis from the geometrical model the property of the FEM software must be taken into account.

If there are some sources for the non-linearity in the model then the only possibility is to analyze all the loading combinations separately. Eurocode 3 [20] allows the use of the proportional loading also in the non-linear case so the loading factors in the loading combinations are constant. The loading combinations are made by combining the basic loading cases and using the load factors defined in the local codes. One loading combination can only be one loading case, if the superposition of loading cases is used. In this case the post-processing program calculates the real loading combinations. Therefore, the serial_number_of_combin_of_loading_case is sufficient information at this stage when generating the structural analysis. This serial number run through all the combinations of the loading cases. These combinations can be the final loading combinations or just the loading cases, depending on the software used for the analysis. One combination of loading cases produces at least one ANALYSIS_CASE.

```
ENTITY ANALYSIS_TYP;
       analysis_type_id : identifier;
       analysis_type_assumptions : OPTIONAL description;
       analysis_type_number : INTEGER;
       type_of_analysis : ANALYSIS_TYPE;
       number_of_eigenvalues : OPTIONAL INTEGER;
  WHERE
WR1 : NOT(type_of_analysis=BUCKLING AND NOT EXISTS
(number_of_eigenvalues));
WR2 :NOT(NOT(type_of_analysis=BUCKLING) AND
EXISTS(number_of_eigenvalues));
  UNIQUE
       UR1 : analysis_type_id;
END_ENTITY;
```

The number of eigenvalues (and corresponding eigenmodes) must be large enough for the calculation of the proper buckling modes for the dimensioning purposes of the skeleton [2]. The use of the first eigenvalue (as in [21]) for the dimensioning of all the members of a steel skeleton usually leads to conservative results. The eigenvalues are needed when calculating the buckling lengths of members, when calculating the lateral buckling load factor of the individual member or when calculating the factors of sway frames according to e.g. Eurocode 3 [20]. So far, the linear static analysis and the linear eigenvalue problems (linear instability, use of geometrical or initial stiffness matrices) have been handled.

```
ENTITY FEA_ENTITY
  ABSTRACT SUPERTYPE OF (ONEOF(ELEMENT,NODE,ANALYSIS_MODEL));
      fea_entity_id : identifier;
      serial_number_of_fea_entity : INTEGER;
      origin : SET [1:?] OF STRING;
      local_source : STRING;
  UNIQUE
      UR1 : fea_entity_id, serial_number;
END_ENTITY;
```

This entity is used as an abstract supertype for entities seen in the definition. The origin is the unique reference to the geometrical model and its objects. Note, that the same instance of the geometrical model may occur in many analysis models (e.g. a corner column in two plane models). Also, the special superelements can be built from the geometrical model. In practice the reference to the geometrical object is done in this study using the complete identification path of the object's identification. The local_source refers to the local model of the geometrical entity given by the origin attribute. For example, one bar or joint (in the geometrical model referred to above) can be divided into five finite elements by the KBS. The string local_source is structured in this case as 'LOCAL_NUMBER_5'.

```
ENTITY ANALYSIS_MODEL
  SUBTYPE OF (FEA_ENTITY);
      model_description : OPTIONAL description;
      model_coord_sys : COORD_SYSTEM_TYPE;
      total_no_nodes : INTEGER;
      total_no_elements : INTEGER;
END_ENTITY;
```

FINITE ELEMENTS

```
ENTITY ELEMENT
  SUBTYPE OF (FEA_ENTITY);
      element_label : ELEMENT_LABEL_TYPE;
      element_typ : ELEMENT_TYPE;
      n : NUMBER_OF_NODES;
      element_material : MATERIAL;
      element_property : PROPERTY;
      element_orientation : ARRAY [1:3] OF REAL;
      parent_model : ANALYSIS_MODEL;
  INVERSE
connectivity : SET [0:?] OF ELT_NODE_CONNECTIVITY FOR connecting_elt;
  WHERE
      connectivity_n : SIZEOF (connectivity) = n;
END_ENTITY;


TYPE ELEMENT_LABEL_TYPE = ENUMERATION OF (MID_BAR, JOINT);
END_TYPE;
```

So far, the element label can have, only two values 'MID_BAR' or 'JOINT'. This information implicates the origin of the element. The skeleton consists of (roughly speaking) joints and bars between them (MID_BARs). This attribute is used when transferring the data to the dimensioning programs of bars or joints. The same information (either the element belongs to MID_BAR or to JOINT) can be found from the origin of the ELEMENT. The element types available (at this stage) are given in the element type definition. Also the MATERIAL and PROPERTY entities must be defined.

Description of how the element's local axis system is oriented with respect to the global axis system of the analysis model is defined using the attribute element_orientation. The orientation is unique, if the local x-axis of the element is from the first node (and it is supposed that the origin of the local system is at the "start end") to the second (it is supposed that the local x-axis of the element is directed from the first node to the second node) node of the element (see entity ELT_NODE_CONNECTIVITY, attribute elt_node_con_number, the entity ELT_NODE_CONNECTIVITY is used also in CIMsteel schema, [9]) and the direction cosines (three, v_x, v_y, v_z) of the local y-axis with respect to the global system are given by this attribute. The third direction cosines of the local coordinate system can be calculated using this information as long as the right handed Cartesian coordinate system is used. If the used FEM program should shift the origin of the element's local coordinate system to the mid-point of the element or somewhere else then the corresponding transformation must be done in the conversion program from this model to the FEM program.

```
TYPE ELEMENT_TYPE = ENUMERATION OF (SPRING, BERN2D, TIMO2D, TAPERED2D,
LAYE2D, BERN3D, TIMO3D, VLAS3D, DIST3D, PL4N2D, BRICK);
END_TYPE;
```

The element types used (at this stage) are given in this definition. The SPRING element is 0-dimensional (no length) spring with two nodes (which may have the same coordinates) and which may have one or more degrees of freedom per node and the spring may have a direction as an attribute. If the nodes of the SPRING element have the same coordinates then the direction must be given. The stiffness matrix of the spring element is diagonal. The spring elements are mainly used when modeling joints. The stiffness of the joint may depend on e.g. the loadings of the joint [20] and then the problem becomes non-linear and iteration is needed. In order to do this iteration, the loading schema is needed.

The BERN2D element is a Bernoulli-Euler-type beam element with two nodes and three DOFs per node and this element is well-known and widely used in the practice. The TIMO2D is the corresponding Timoshenko beam element. This element with linear interpolations of displacements between the nodes is used as an interface element between e.g. beam and plate models [16]. The explicit stiffness matrices for tapered beam elements are available in the literature [22, 23] and are recommended to be used when available. The LAYE2D element is a two dimensional layered beam element which is used for composite or Sandwich beams [24]. The element has two nodes at the ends of the element and four DOFs per node and the exact interpolation functions for the displacements (which fullfil the corresponding differential equations) are used between the nodes. The Sandwich beam can have thin or thick faces. There are two displacement and two rotation DOFs at each node.

The BERN3D and TIMO3D elements are the well-known three dimensional beam elements (six DOFs per node). The VLAS3D is a three dimensional beam element including warping (seven DOFs per node). When using this element it is possible to use the exact interpolation in the linear case [25] if the corresponding stiffness matrices are available in the analysis program. The stiffness matrices based on an approximative cubic interpolation polynimials between the nodes have been known for a long time [26] and they have been installed in many commercial programs (such as ABAQUS). All the elements may not be available in the analysis program of the designer. This information (the availability of the element types) is asked for the designer by the KBS. The VLAS3D element is of the most importance when analyzing steel skeletons. This element can be used for the calculation of e.g. the lateral torsional or torsional buckling load factor of the member which is needed when dimensioning the member according to Eurocode 3 [20].

The next element, DIST3D, is perhaps not available in many analysis programs, because it is so new [27]. This element is developed especially for the analysis of thin walled open purlins widely used in steel skeletons. This element is the expansion of the VLAS3D element so that there are nine, eleven or more DOFs per node. The pairs of extra DOFs (compared to the VLAS3D element) are generated due to the distortion of the profile and the pairs always include the rotation and the corresponding warping DOF. This element can be used (so far) only for the linear analysis because there exists no geometrical (initial stiffness) matrix for this element needed in the buckling analysis. The PL4N2D element is a four noded plane stress element with bubble modes. This element has been used when more detailed analysis model is needed, especially for joints. The BRICK element is a 3D brick element with 21 DOFs per element and it must be used with care in order to avoid too heavy models. The other element types (such as thin or thick shell elements) are easy to add to the model.

The KBS gives some propositions for the use of different elements in the analysis. These are e.g. the number of beam elements when calculating the buckling load factor. The aspect ratios and similar rules for e.g., the PLAN2D elements are programmed to the KBS when choosing the local models for different joints. These rules are mainly taken from the literature [28].

MATERIAL

```
ENTITY MATERIAL;
```

```
        material_id : identifier;
        material_name : STRING;
        material_standard : STRING;
        elastic_modulus : REAL;
        poisson_ratio : REAL;
        density : OPTIONAL REAL;
        thermal_expans : OPTIONAL REAL;
        yield_strength : OPTIONAL REAL;
        ultimate_strength : OPTIONAL REAL;
        ultimate_strain : OPTIONAL REAL;
END_ENTITY;
```

The material entity is about the same as in [8] without the material safety factor. It must be noted that e.g. the yield strength of the material is dependent on the thickness of the geometrical entity to which this material entity is applied. This information (the thickness) must be included in the conversion from the geometrical model to the analysis model. Also the ultimate strain is of main importance when evaluating the data. The use of Eurocode 3 [20] for dimensioning is allowed only if this strain value is larger than 20%.

When applying the proposed model to the structural analysis of other than steel skeleton (wood, concrete, composite) then the material entity needs the most modifications. Other entities of the present model fit rather well for the analysis of those skeletal structures.

ELEMENT-NODE CONNECTIVITY

```
ENTITY ELT_NODE_CONNECTIVITY;
        elt_node_con_number : INTEGER;
        connecting_elt : ELEMENT;
        connecting_node : NODE;
        fixity : RELEASE;
  UNIQUE
        UR1 : elt_node_con_number, connecting_node, connecting_elt;
END_ENTITY;
```

This entity is also used in [9] but only with BERN2D elements.. If the node is at the "start end" of the element, then the elt_node_con_number = 1; similarly, for a 1 dimensional element, if the node is at the "end end" of the element, then the elt_node_con_number = 2. The previous is true for 1 dimensional (beam) element with two nodes. If there are more nodes (e.g. 3 nodes in 1 dimensional element) then the elt_node_con_number have values 1, 2, 3 where 1 corresponds to the "start end", 2 corresponds to the mid-node and 3 corresponds to the "end end" and so on. The local coordinate system is defined here. So, this attribute includes the order of nodes in an element. Fixity declares the instance of RELEASE associated with this elt_node_connectivity. Note, that all the entities ELT_NODE_CONNECTIVITY must have this attribute. The connectivity defined here is different to that often used in FEM codes [29] but the information is the same. Note also, that only the finite element methods for the displacement method are used in this study. The DOFs as second derivatives are omitted here. They are proposed to DOFs in --[12] because they are needed e.g. in the mixed or hybrid methods.

NODES

```
ENTITY NODE
  SUBTYPE OF (FEA_ENTITY);
      node_label : NODE_LABEL_TYPE;
      node_coords : POINT;                          --In global
system
      restraints : OPTIONAL BOUNDARY_CONDITION;
      parent_model :  ANALYSIS_MODEL;
END_ENTITY;

TYPE NODE_LABEL_TYPE = ENUMERATION OF (MID_BAR, JOINT,
            MID_BAR_AND_JOINT);
END_TYPE;
```

The label of node is similar to that of element but the node can belong to both MID_BAR and JOINT.

BOUNDARY CONDITIONS

```
ENTITY BOUNDARY_CONDITION;
       boundary_cond_number : INTEGER;
       boundary_condition_label : OPTIONAL label;
       boundary_cond_description : OPTIONAL description;
       bc_x_rotation : OPTIONAL LIST [1:?] OF REAL;
       bc_y_rotation : OPTIONAL LIST [1:2] OF REAL;
       bc_z_rotation : OPTIONAL LIST [1:2] OF REAL;
       bc_x_displacement : OPTIONAL REAL;
       bc_y_displacement : OPTIONAL REAL;
       bc_z_displacement : OPTIONAL REAL;
       x_skew_angle : OPTIONAL REAL;
       y_skew_angle : OPTIONAL REAL;
       z_skew_angle : OPTIONAL REAL;
       warping : OPTIONAL LIST [1:?] OF REAL;
  UNIQUE
       UR1 : boundary_cond_number;
END_ENTITY;
```

The same coding system as in [9] is used for giving the boundary conditions. If the value of a generalized displacement (translation, rotation or warping) is 0 (zero) then the corresponding DOF is free (no load transfer). If the value is -1 then the DOF is fixed. If there is some positive value then it is understood to be the spring stiffness for this DOF and the number of elements and nodes (if spring elements are used, see also entity release) must be updated automatically. Note, that the given value e.g. for the attribute bc_x_rotation does not mean a predefined generalized displacement. If there are some predefined generalized displacements, then they are given in the loading schema.

Note, also, that usually e.g. base joints are modelled to behave elastically and that the real joints may behave in some other way. Usually, there are two main sources for the non-linear behaviour of the joints of steel skeletons: contact problem and material non-linearity due to plastification of the material. The plastification does not occur at the loading level where the joints are to be loaded during their life time, instead the contact problem (often combined with the prying effect) occurs also at that load level. In many practical joints a two dimensional local model near joints combined with the solution of the contact problem is enough for the practice. Examples for base bolt joints are given in [30]. Similar use of local profound model for joints combined with beam elements for bars between joints has been recommended for other joints [31, 32, 33].

The boundary condition (and release) entities are generated usually from the joints in this study. The limit between different structural assemblies is sometimes not clear. In this study, only the steel skeleton and its assemblies are handled. The analysis of base bolt joints does not include here the analysis of concrete footing or soil below the joint, which may have signifiCiant effect on the total behaviour of the steel skeleton [2]. The analysis model of the skeleton must be interconnected to the total model of the building (including the soil below the building) in the future. This integration will produce the loadings to the steel skeleton in the future.

```
ENTITY RELEASE;
      release_number : INTEGER;
      release_label : OPTIONAL label;
      release_description : OPTIONAL description;
      release_axial_force : OPTIONAL REAL;
      release_y_force : OPTIONAL LIST [1:2] OF REAL;
      release_z_force : OPTIONAL LIST [1:2] OF REAL;
      release_torsional_moment : OPTIONAL LIST [1:?] OF REAL;
      release_y_bending_moment : OPTIONAL REAL;
      release_z_bending_moment : OPTIONAL REAL;
      x_skew_angle : OPTIONAL REAL;
      y_skew_angle : OPTIONAL REAL;
      z_skew_angle : OPTIONAL REAL;
      release_bimoment : OPTIONAL LIST [1:?] OF REAL;
  UNIQUE
      UR1 : release_number;
END_ENTITY;
```

The same coding system (for generalized forces) as with the boundary condition entity (for generalized displacements) is used here (and in [9]). When using a zero value for some generalized force it is usual that there is a special element installed to the application software. The most common case when using beam elements is the case, where there is a hinge at one or at both ends of the element. When using positive values for some generalized forces then the positive values mean the corresponding spring stiffnesses, not given forces. In this case the spring elements and the corresponding nodes must be generated automatically (as for the boundary condition entity) or some softwares include so called equation or MPC (multi point constraint) possibilities, which can be used to give linear relationships for DOFs. The conversion program for the analysis program's input file must take care of this. When analyzing a steel skeleton there may exist joints which must be analyzed as semirigid including non-linear stiffness properties (see e.g. [34]). In this case the iteration is needed. The most common attribute values for the generalized forces within this entity are -1, which means than the connection between element's local node to the node is fixed (absolutely rigid).

CONCLUSIONS

The product data model for the structural analysis of steel frames is proposed in the paper. The model seems to be rather simple, yet detailed enough for the practical applications. The supporting Knowledge Based System (KBS) between geometrical model and the analysis model must be developed towards practical cases in order to apply the proposed models in real life projects. The schemata for loading and results models must be developed in order to apply the models effectively in the complete design process. Also, the direct links to the dimensioning of joints and bars between them must be developed. There are many projects at the moment where the software tools for these tasks are being developed.

In conclusion, parts of the design process and data exchange between them can now be done (more or less) automatically by the computer without extra work for the structural designer. The designer must only answer some guestions made by the KBS. The process starts from the geometrical model made by using a CAD program and the analysis model following the schema given in this paper is generated by the KBS (also used for selection of joints).

The proposed product model for the structural analysis of skeletal structures seems rather well suited to the analysis of other than steel structures also. The entity MATERIAL needs the most modifications when applying the proposed model e.g. for wooden structures. Also, the geometrical model (here the FST-model was used) needs some modifications in order to be used for e.g. wooden structures. The analysis-schema has been checked using the ECCO Tool Kit [35]. The Tool Kit did not find any lexical errors in the schema.

ACKNOWLEDGEMENTS

The financial support of the Finnish Academy is gratefully acknowledged. Finnish Constructional Steelwork Association (FCSA) is acknowledged for the possibility to use the

CIMsteel material. Special thanks are due to Mr. Antero Miettinen for his help in writing the programs.

REFERENCES

1. Mikkola M., Computer Aided Design of Wood Trusses Using Nail Plates, Proceedings of CIVIL-COMP 85, Volume 1, CIVIL-COMP PRESS, Edinburgh, 149-154 (1985).

2. Heinisuo M., Möttönen A., Paloniemi T., Nevalainen P., Automatic design of steel frames in a CAD-system, Proceedings of the 4th Finnish Mechanics Days, Ed. Niemi E., Research Papers 17, Lappeenranta University of Technology, Lappeenranta, 197-204 (1991).

3. Arabshahi S., Barton D.C., Shaw N.K., Steps towards CAD-FEA integration, Engineering with Computers 9, 17-26 (1993).

4. Brun J.M., Afzali M., Serrafero P., Neumann A., Integrated product design and manufacturing system - Esprit project n°2590, Proceedings of MICAD 92 International Conference, Paris, 561-575 (1992).

5. Remondini L., Leon J.C., Trompette P., Towards an integrated architecture for the structural analysis of mechnaical structures, Information Processing in Civil and Structural Engineering Design, B. Kumar (Editor), CIVIL-COMP Ltd, Edinburgh, Scotland, 65-73 (1996).

6. Turkiyyah G.M., Fenves S.J., Knowledge-Based Assistance for Finite-Element Modeling, IEEE EXPERT INTELLIGENT SYSTEMS & THEIR APPLICATIONS, Vol. 11, N. 3, 23-32 (1996).

7. Aas, J., Data Exchange in Finite Element Analysis - a Functional Requirement Specification, NAFEMS, Ref R0005 (1992).

8. Haller H.-W., Ein Produktmodell für den Stahlbau, Dissertation, Bauingenieur- und Vermessungswesen der Universität Friedericiana zu Karlsruhe (TH), Karlsruhe, (1994).

9. CIMsteel: Integration Standards (Release One), University of Leeds, Department of Civil Engineering, (1995).

10. ISO 10303-11: Industrial Automation Systems - Product Data Representation and Exchange, Part 11: The EXPRESS language reference manual, (1992).

11. ISO 10303-21: Industrial Automation Systems - Product Data Representation and Exchange, Part 21: Clear text encoding of the exchange structure, (1993).

12. ISO 10303-104/CD: - FEA Reference Model, ISO TC184/SC4/WG3/P9, Document N 21, 3 January, (1991).

13. Boender E., Bronsvoort W.F., Post F.H., Finite-element mesh generation from constructive-solid-geometry models, Computer-Aided Design, Volume 26, Number 5 May, 379-392 (1994).

14. Romeijn A., Puthli R., Wardenier J., Guidelines on the numerical determination of stress concentration factors of tubular joints, Tubular Structures V, Eds. M.G. Coutie, G. Davies, E & FN SPON, London, 625-639 (1993).

15. Heinisuo M., Expert Systems for Design of Steel Structures, Part 3: Expert rules for design of bars and joints, Tampere University of Technology, Department of Civil Engineering, Structural Mechanics, Report 16, Tampere, (1994).

16. Heinisuo M., Hyvärinen J., Knowledge based approach to the design of structural steel joints, Information Processing in Civil and Structural Engineering Design, B. Kumar (Editor), CIVIL-COMP Ltd, Edinburgh, Scotland, 75-83 (1996).

17. Heinisuo M., Hyvärinen J., Steel joint analysis using product model, Eight Nordic Seminar on Computational Mechanics, Chalmers University of Technology, Publication 95:15, Eds. R. Larsson, N.-E. Wiberg, 1-4 (1995).

18. Heinisuo M., Rautakorpi J., Generating Product Model for Structural Analysis of the Steel Skeleton from the Product Model of the Geometry of the Skeleton, Tampere University of technology, Department of Civil Engineering, Structural Mechanics, Report 25, Tampere, (1998), (in Finnish).

19. Hyvärinen J., Expert Systems for Design of Steel Structures, Part 2: Hierarchical aggregation form of steel skeleton, Tampere University of Technology, Department of Civil Engineering, Structural Mechanics, Report 15, Tampere, (1994).

20. Eurocode 3: Design of steel structures. Part 1.1. General rules and rules for buildings, ENV-1993-1-1, Comission of the European Communities, European prenorm, Brussels, Belgium, (1992).

21. Yang Y.-B., Kuo C.-C., Automatic Design System for Steel Frames, CIVIL-COMP 89, CIVIL-COMP PRESS, Edinburgh, (1989).

22. Eisenberger M., Explicit stiffness matrices for non-prismatic members, Computers & Structures 20 715-720 (1985)4.

23. Banerjee J., Williams F., Exact Bernoulli-Euler static stiffness matrix for a range of tapered beam-columns, International Journal for Numerical Methods in Engineering 23, 1615-1628 (1986).

24. Heinisuo M., Exact Finite Element Method for Layered Beams, Tampere University of Technology, Publications 56, Tampere, (1989).

25. Annila A., Stability of open thin walled beams by finite element method, Diploma thesis, Tampere University of Technology, Tampere (1988), (in Finnish).

26. Barsoum R.W., Gallagher R.H., Finite element analysis of torsional and torsional-flexural stability problems, International Journal of Numerical Methods in Engineering, 2, 335-352 (1970).

27. Heinisuo M., Liukkonen V.-P., Tuomala M., New beam element including distortion, Proceedings Volume 1, Nordic Steel Construction Conference '95, Swedish Institute of Steel Construction, Malmö, Sweden, 65-72 (1995).

28. Kardestuncer H., (Ed.): Finite Element Handbook, McGraw-Hill, New York, (1987).

29. Hughes T.J.R., The Finite Element Method, Linear Static and Dynamic Finite Element Analysis, Prentice-Hall, Inc, Englewoods Cliffs, New Jersey, (1987).

30. Heinisuo M., Raiskila M., Modeling of a joint between steel column and concrete footing, Teräsrakenne 4 30-34 (1986) (in Finnish).

31. Souissi R., Flexibility of tubular joints, Tubular Structures, Eds. Niemi E., Mäkeläinen P., Elsevier Applied Science, Essex, 137-146 (1990).

32. Efthymiou M., Local rotational stiffness of unstiffened tubular joints, Koninklijke/Shell Exploratic in Produktic laboratorium, Rilswijk, report no RKER 85.199, (1985).

33. Coutie M., Davies G., Philiastides A., Roodbaraky K., Yeomans N., Fully overlapped RHS joints - isolated joints and trusses, Tubular Structures, Eds. Niemi E., Mäkeläinen P., Elsevier Applied Science, Essex, 1-10 (1990).

34. Richard R.M., Abbot B.J., Versatile Elastic-Plastic Stress-Strain Formula, ASCE Journal of the Engineering Mechanics, 101:No. EM4, 511-515 (1975).

35. ECCO Tool Kit, An Environment for the Evaluation of EXPRESS Models and the Development of STEP based IT Applications, http://www-rpk.mach.uni-karlsruhe.de/public/Mitarbeiter/Staub/Ecco

Markku Heinisuo, Dr. Tech.
KPM-Engineering Oy
Yliopistonkatu 60 C
33100 Tampere, Finland
email: markku.heinisuo@kpmeng.fi

Juha Hyvärinen, M.Sc. (Eng)
VTT Building Technology
P. O. Box 1801
02044 VTT, Finland
email: juha.hyvarinen@vtt.fi

Mika Salonen, M.Sc. (Eng)
VTT Building Technology
P. O. Box 1801
02044 VTT, Finland
email: mika.salonen@vtt.fi