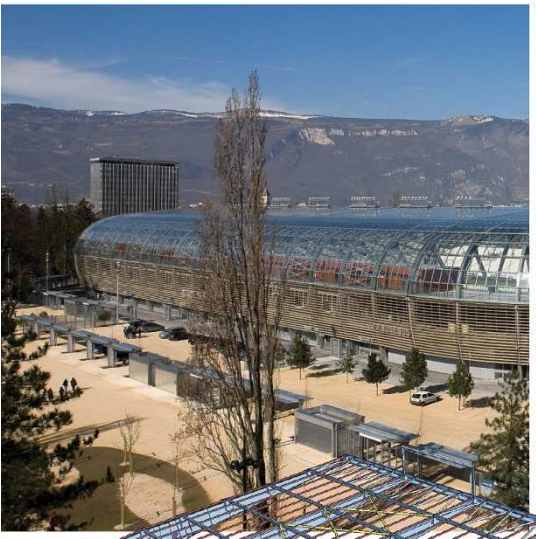
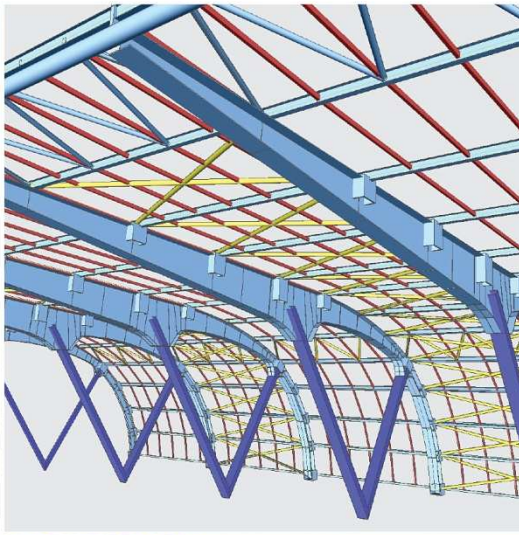
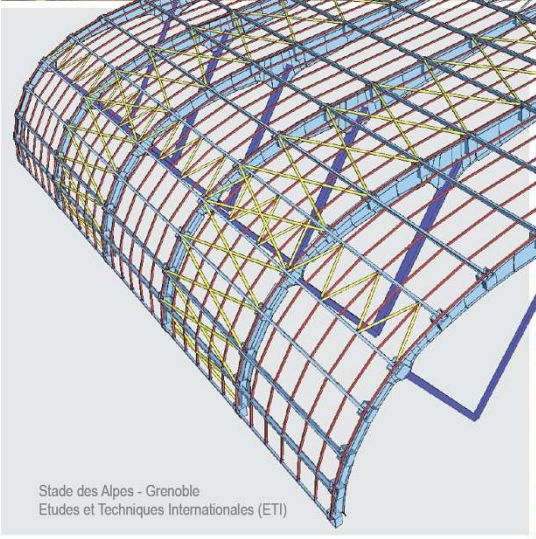
NEMETSCHEK Scia







Stade des Alpes - Grenoble
Etudes et Techniques Internationales (ETI)

Tutorial – End+continue in commands

How to use End+continue for different types of commands

Scia Engineer

Release: Scia Engineer 2011

The information contained in this document is subject to modification without prior notice. No part of this document may be reproduced, transmitted or stored in a data retrieval system, in part or in total, in any form or by any means, electronic or mechanical, for any purpose without the express permission in written from the publisher. SCIA Software is not liable for any direct or indirect damages resulting from imperfections in the documentation and/or the software.
© Copyright 2009 SCIA Software. All right reserved.

Introduction **4**

How to use End+continue in command for inserting structure **5**

Insert members with different cross-section in one command **5**
Insert plates with different thickness in one command **8**
How to use End+continue during load inserting **11**

Insert loads with different value **11**

Introduction

Each command (e.g. input of a beam, definition of a support, etc.) must be finished and started again when the user wants e.g. to insert a member with different properties than the previous member(s). From now on this is not necessary. It is possible to insert a member with different properties within one run of the command using “End+continue” functionality.

For example:

1) The user starts the command to insert a beam with cross-section CS1
2) “End+continue” is used during the command – the already defined beams are confirmed and the command remains “open”
3) The user changes in properties the cross-section from CS1 to CS2
4) The user continues with the command for beams (with cross-section CS2)
5) As a result, there are beams with CS1 and CS2 created in one command

Advantage of “End+continue”:

- One command for more different entities
- Less clicking
- More fluent work
- Faster work

“End+continue” can be used for different types of inserting commands – structure, lines, dimension lines, member data, etc.

When “End+continue“ button is inactive during inserting then it is not possible to use it in that particular case. This can happen mostly for member data (steel, concrete …) because of complex dialogue and sub-dialogue cooperation.

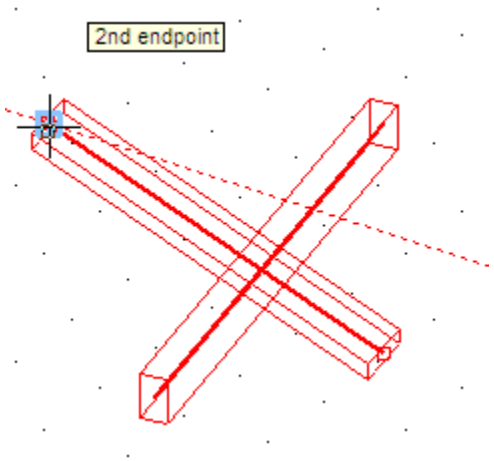
How to use End+continue in command for inserting structure

Insert members with different cross-section in one command

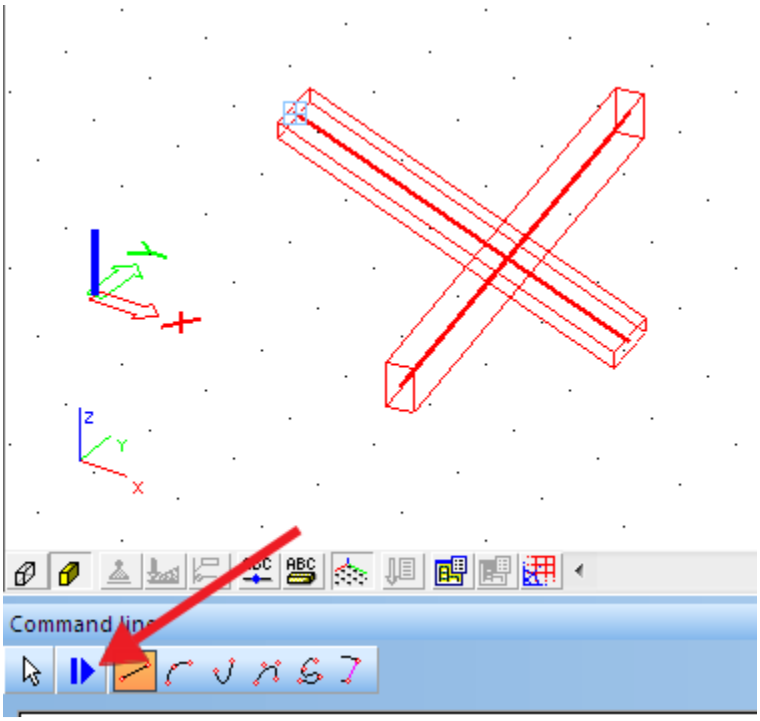
1) Open the project “end+continue.esa”.
2) Switch to the Structural service.
3) Start command for inserting a member with the default properties.

Name	B1
Type	general (0)
Analysis model	Standard
CrossSection	CS1 - Rectangle (500; ...
Alpha [deg]	0,00
Member system-line at	centre
ey [mm]	0
ez [mm]	0
LCS	standard
LCS Rotation [deg]	0,00
FEM type	standard
Buckling and relative lengths	Default
Layer	Layer1 ...

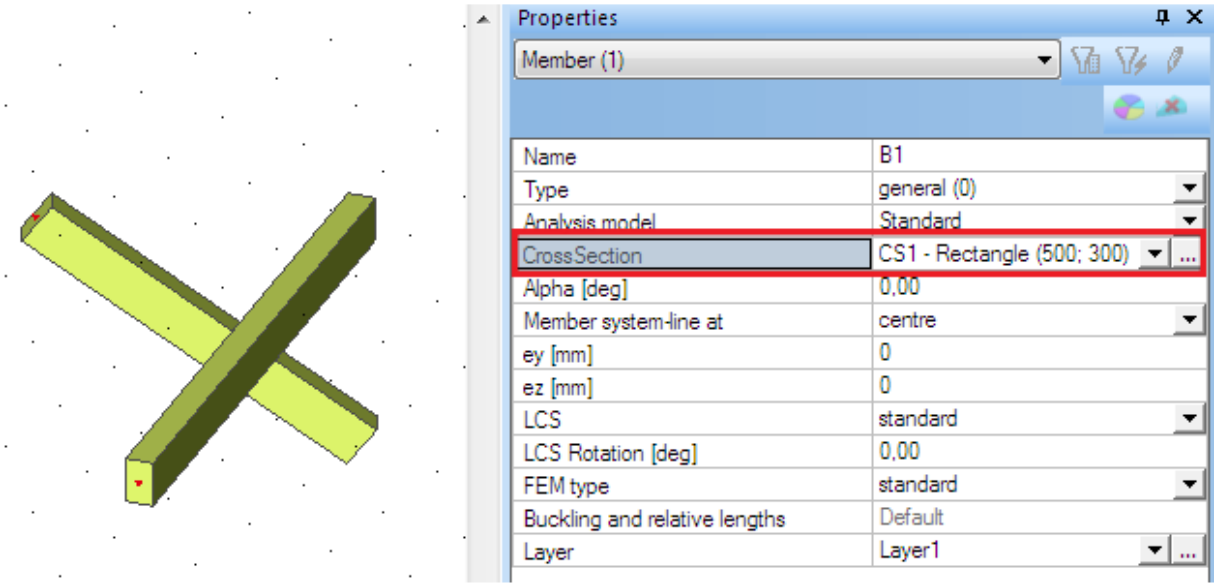
4) Insert two members in the project.



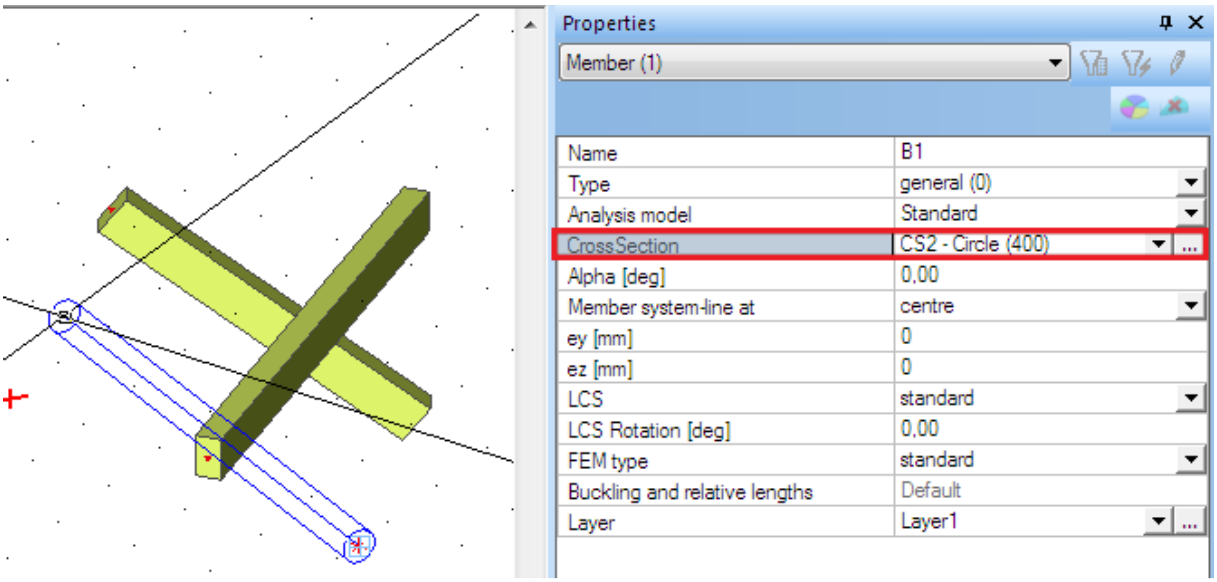
5) Members are still red. Now click the “End+continue“ button on the “fast toolbar” at the bottom of the 3D window. See the picture below.



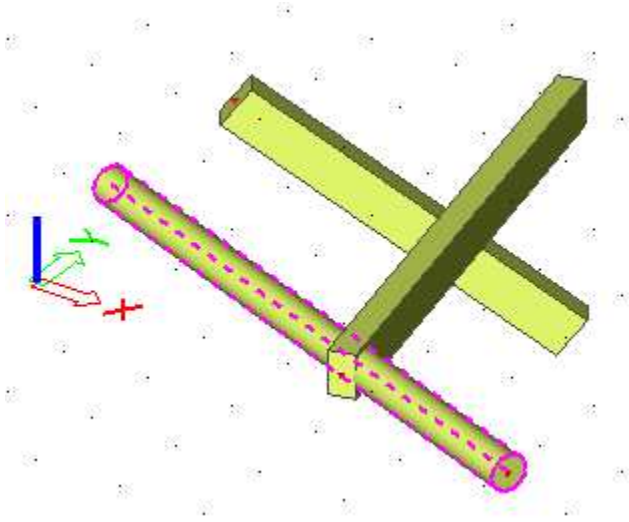
6) Members get their shape and rendered edges (as rendering is switched ON in the “apply.esa” project). They are inserted to the project with CS1. Now it is possible to change the properties. Change the cross-section from CS1 to CS2.



7) Immediately start inserting a new member into the 3D window. The new member is inserted with different cross-section – CS2.



8) End command for input of members by double-ESC. There are 3 members in the project and one has different cross-section.



Note

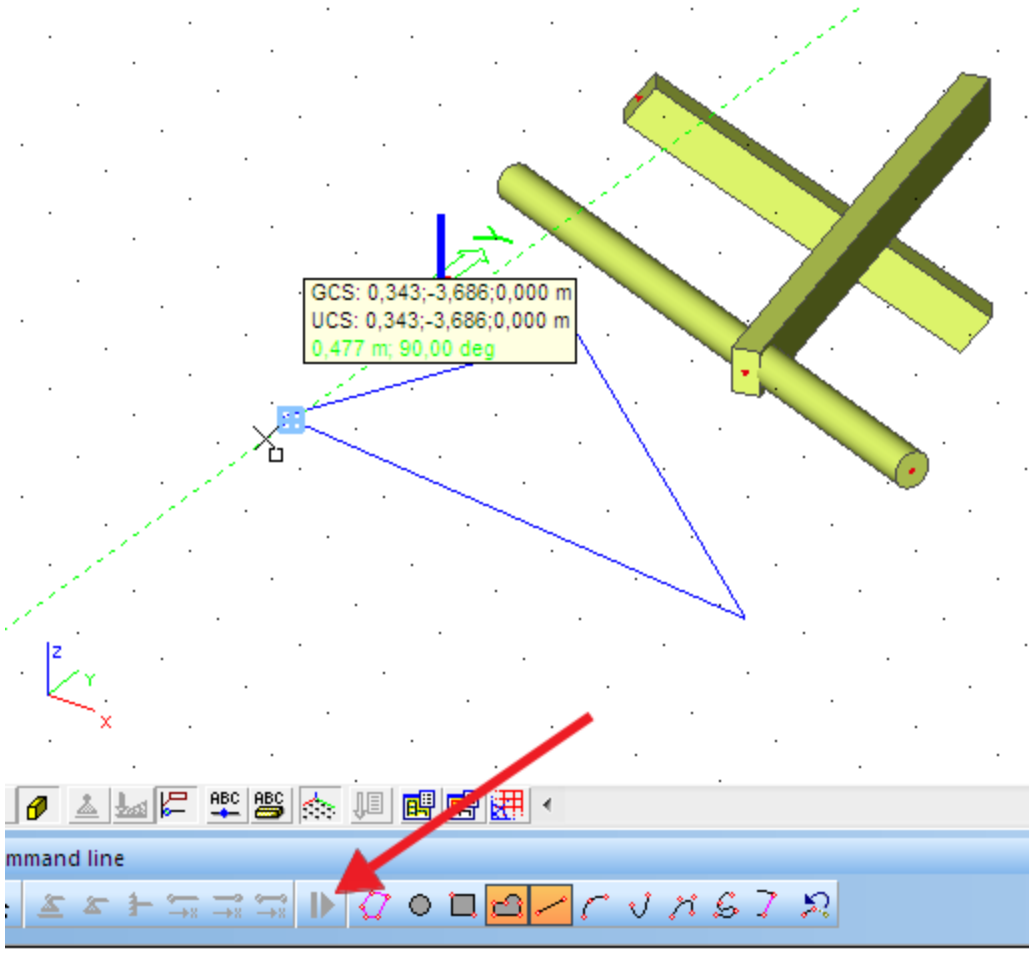
When some properties are changed without using the “End+continue“ button, all the previously inserted members are affected by this change. In our case all members would be inserted with CS2.

Insert plates with different thickness in one command

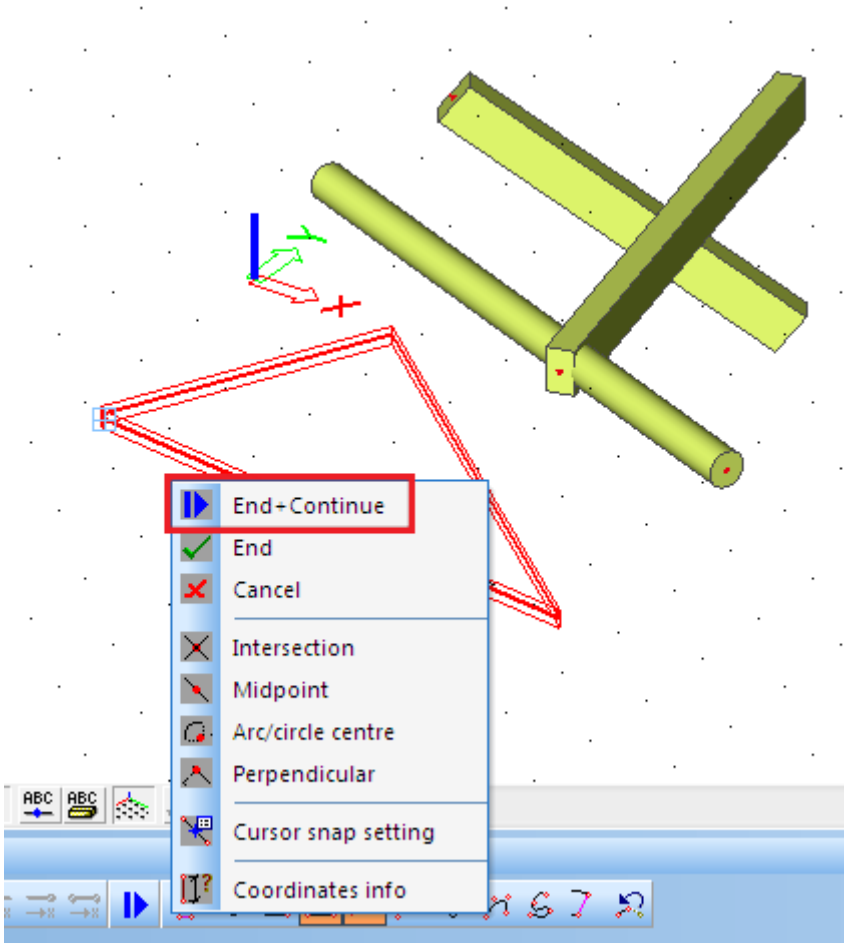
1) Start to insert a plate in the “end+continue.esa” project with the default properties.

Name	S1
Type	plate (90)
Analysis model	Standard
Material	C12/15
FEM model	Isotropic
FEM nonlinear model	none
Thickness type	constant
Thickness [mm]	200
Member system-plane at	centre
Eccentricity z [mm]	0
LCS Type	Standard
Swap orientation	no
LCS Angle [deg]	0,00
Layer	Layer1

2) Insert one plate, notice that during inserting the polygon (polygon has blue colour) the “End+continue“ is button inactive. The “End+continue“ button can be used only after creating a plate (after end polygon command, this can be made by one ESC or the “End” button in the context menu).
The inactive “End+continue” button is shown in the picture below:

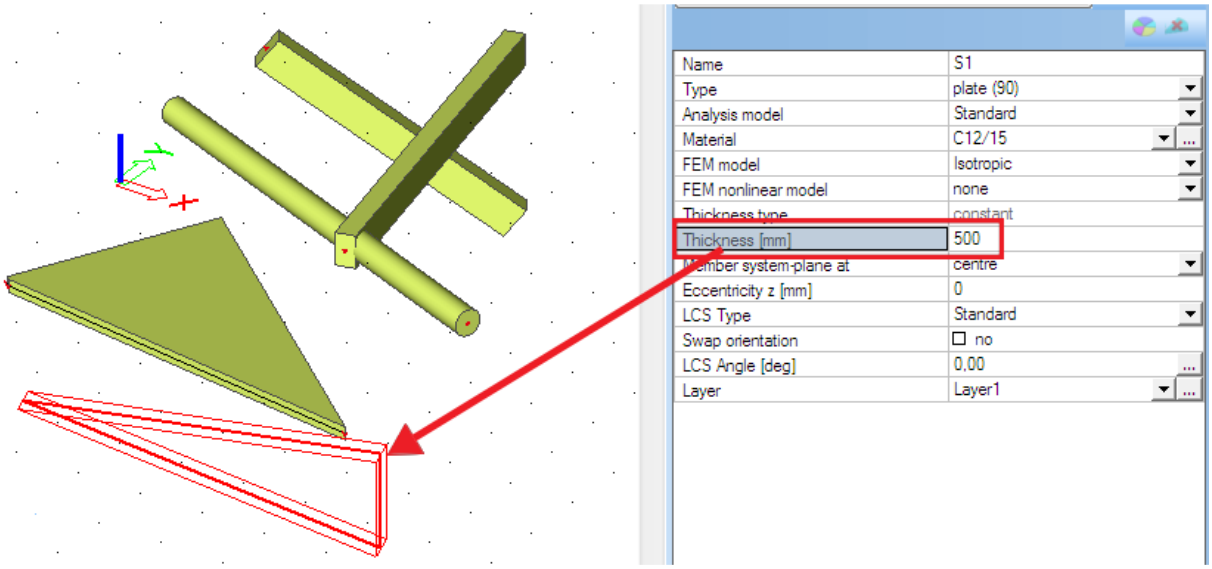


3) End inserting the polygon by one ESC. When the plate is still red, click the “End+continue“ button – this time use context menu.



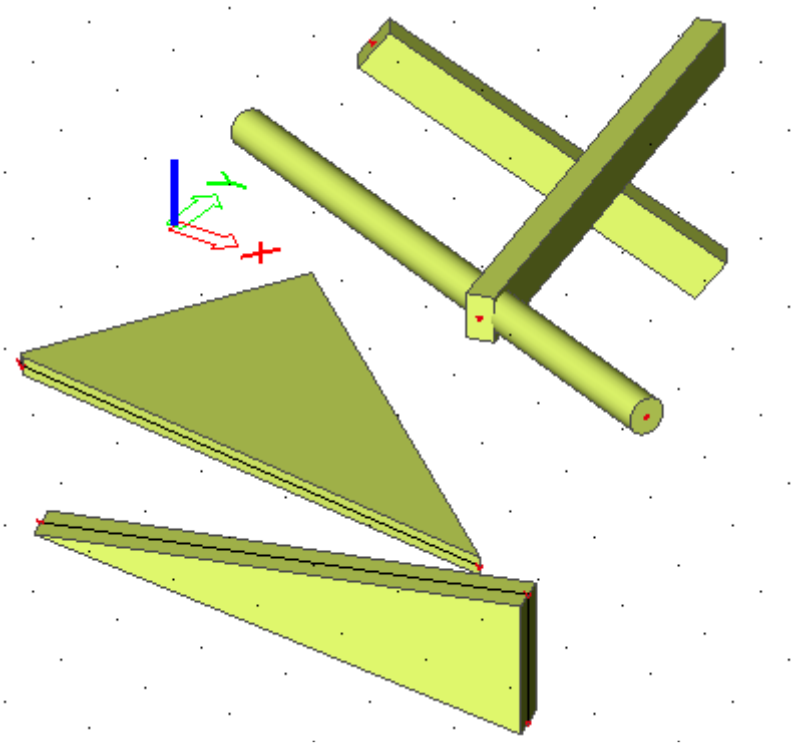
4) The plate is now inserted with the default thickness in the properties. Now it is possible to change the thickness for the next plate.

5) Change thickness from 200mm to 500mm and start immediately inserting the next plate into the 3D window.



6) End inserting by double ESC after creating the second plate.

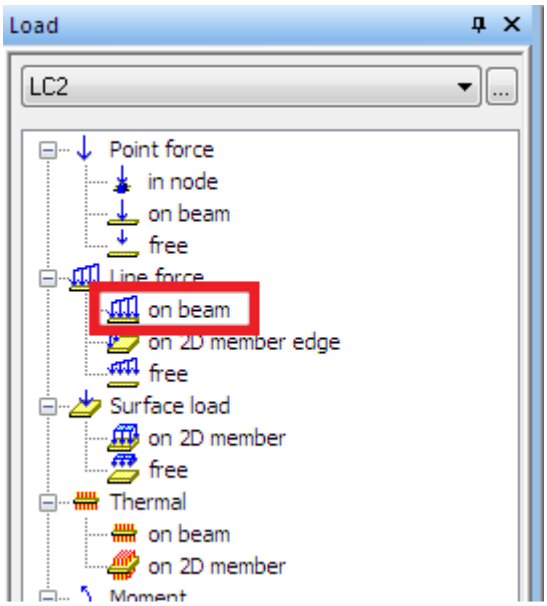
7) The new plate is inserted with the thickness value 500mm.



How to use End+continue during load inserting

Insert loads with different value

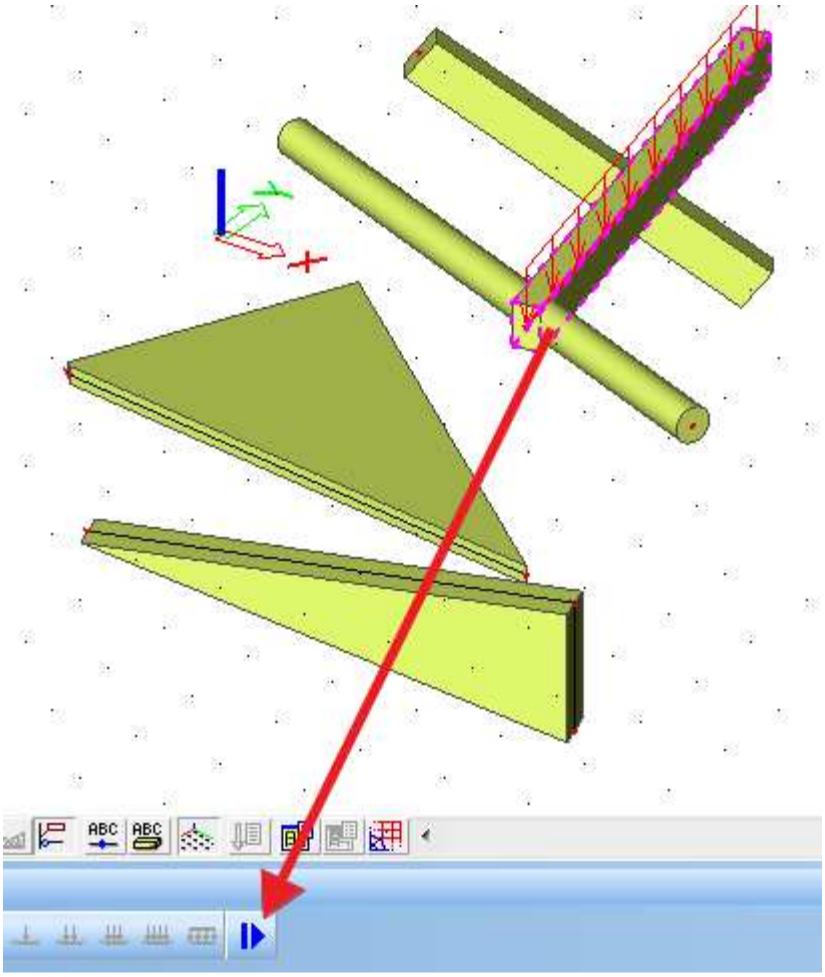
1) Create a new load case in the project.
2) Select function Line force on beam.



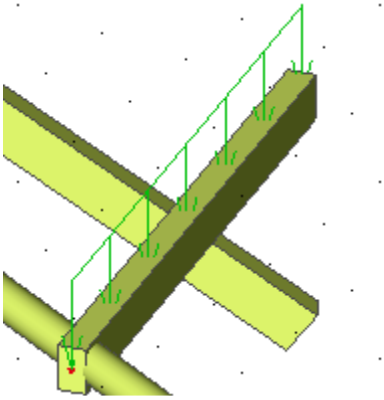
3) Start the command with the default properties. Insert force on beam B1.

Name	LF1
Direction	Z
Type	Force
Angle [deg]	
Distribution	Uniform
Value - P [kN/m]	-1,00
Load above joint	☐ no
Geometry	
System	LCS
Location	Length
Extent	full
Coord. definition	Rela
Position x1	0,000
Position x2	1,000
Origin	From start
Eccentricity	
Eccentricity ey [m]	0,000
Eccentricity ez [m]	0,000

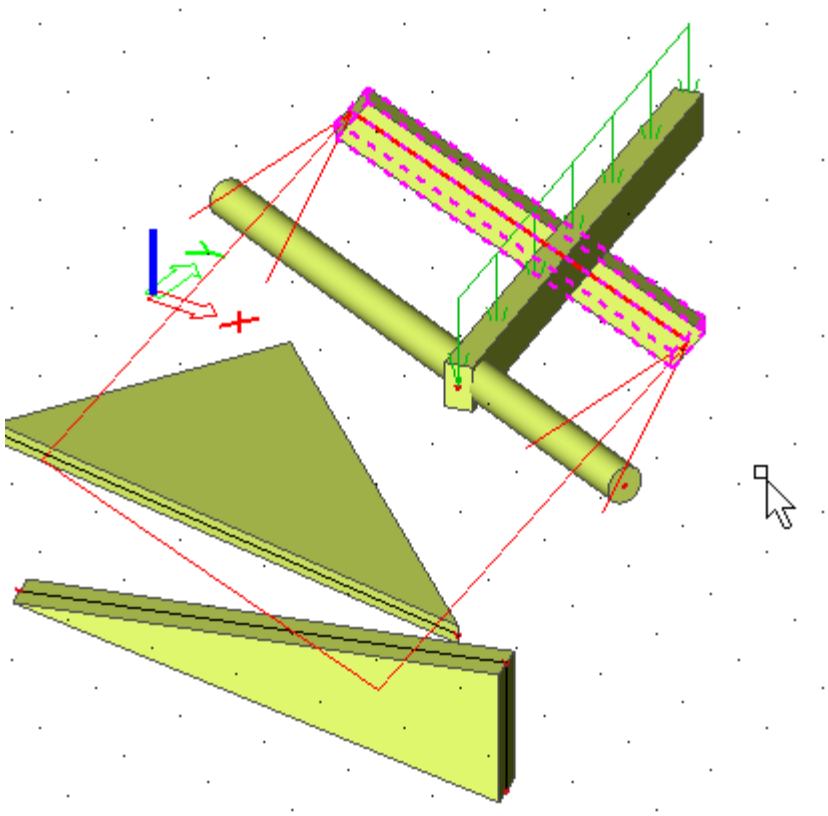
4) When the force is inserted (by selecting B1) and becomes red, the “End+continue“ button is activated.



5) Click the “End+continue“ button. The first force becomes green. Change the force to 5kN. A new force with value 5kN can be inserted.



6) Insert a new force on B2.



7) End the function by one ESC.
8) The whole example is stored in project “end+continue_finished.esa”.