Tutorial

Creating an analytical field to define a varying shell thickness

Preface

The purpose of this tutorial is to show how to create an analytical field to define a varying shell thickness using BRIGADE/Plus.

Contents

1 Overview

A varying shell thickness for a shell structure can be defined by using analytical fields. This tutorial describes the procedure of how to create an analytical field and define a varying shell thickness for the shell structure. The procedure includes the following:

- Create a coordinate system
- Define an analytical field
- Create shell sections
- Assign sections
- Create output field request for shell thickness
- Visualizing the varying shell thickness

2 Create varying shell thickness

2.1 General

The model used throughout this tutorial is a concrete bridge shown in Figure 2-1. A sketch of the varying deck thickness is shown in Figure 2-2. The dotted line shows the reference surface of the shell sections. In Table 1-1 material parameters used for the concrete can be found. An analytical field to define a varying shell thickness will be created for the bridge deck.

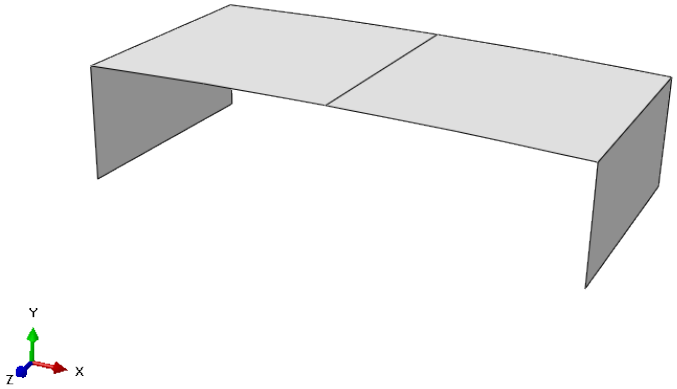


Figure 2-1 Bridge model.

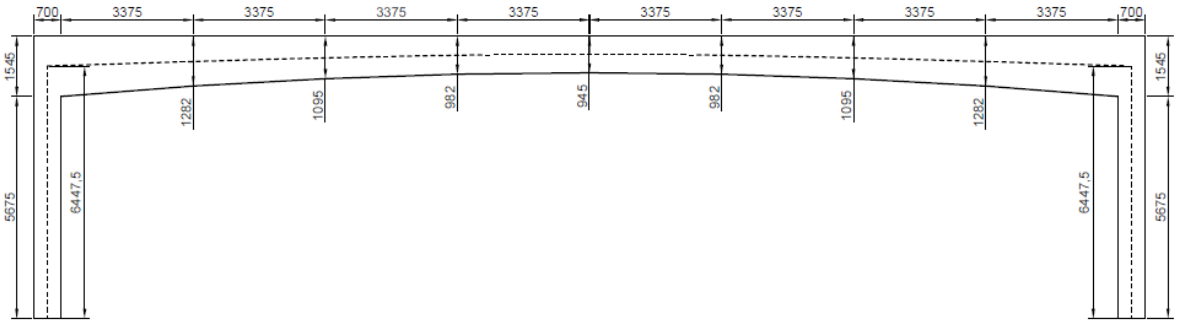


Figure 2-2 Bridge structure

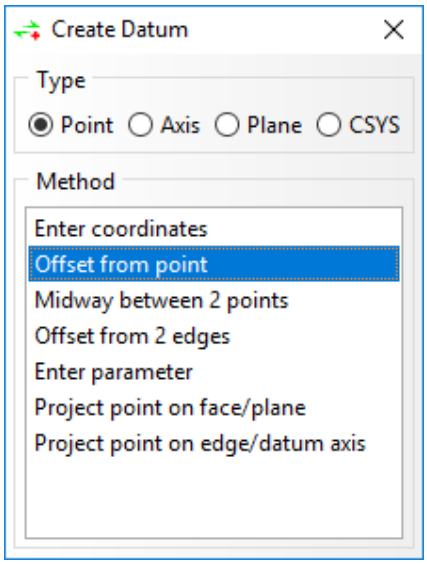
Table 1-1 Concrete parameters.

Symbol	Value	Unit	Description
ρ	2500	kg/m^3	Density
E	35	GPa	Young's modulus
υ	0.2	-	Poisson's ratio
α	1e-5	K^{-1}	Thermal expansion

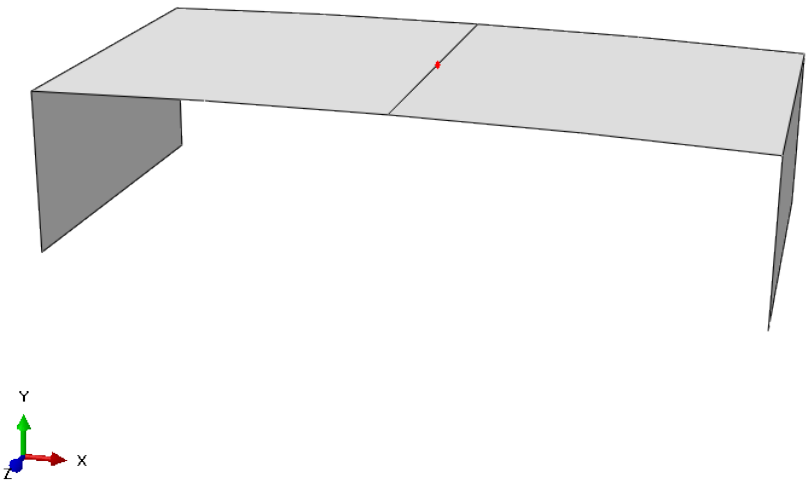
2.2 Create coordinate system

A coordinate system will be created to serve as reference coordinate system for the analytical field. The thickness will vary along the defined axis of the local coordinate system.

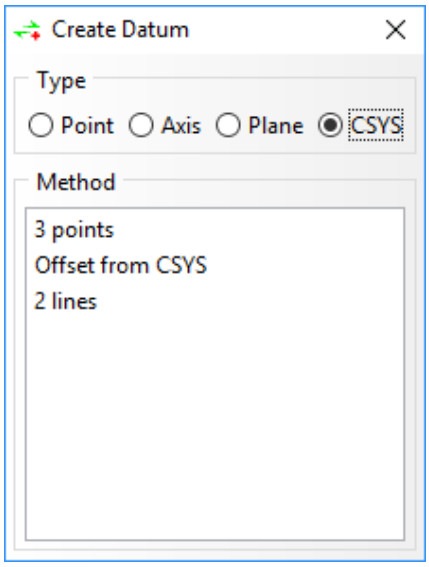
1. Switch to the **Part** module, from the main menu select **Tools → Datum**. In the **Create Datum** dialog set **Type** to **Point** and choose **Offset from point** in the **Method** list.



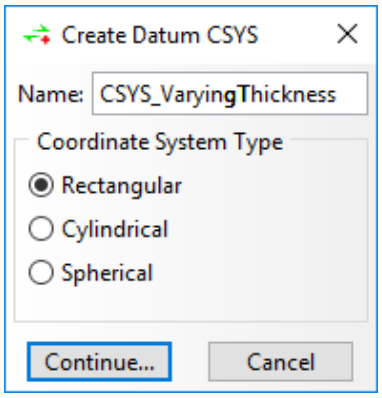
2. Select the midpoint of the deck in the viewport and enter (0.0,1.0,0.0) in the prompt area. Press enter on the keyboard to proceed.



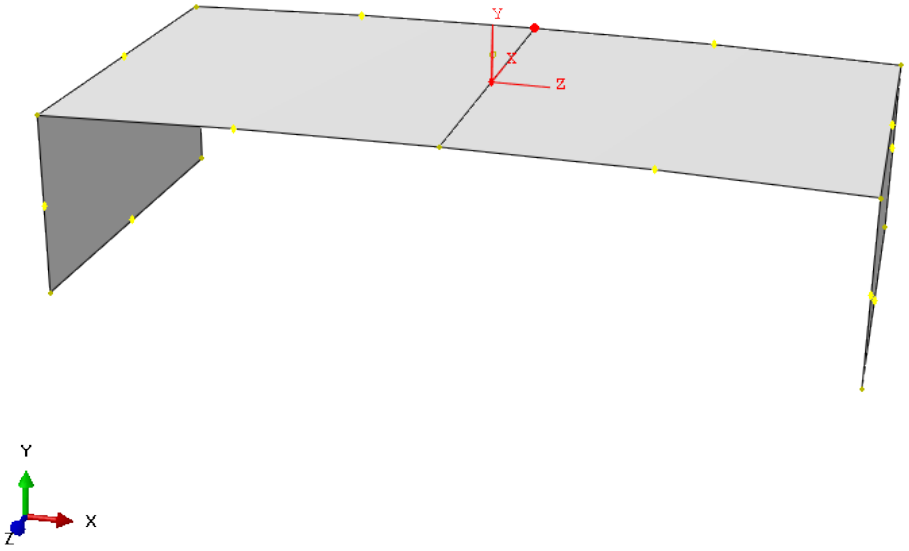
3. The next step is to create the coordinate system. From the main menu select **Tools → Datum**. In the **Create Datum** dialog set **Type** to **CSYS** and choose **3 points** in the **Method** list.



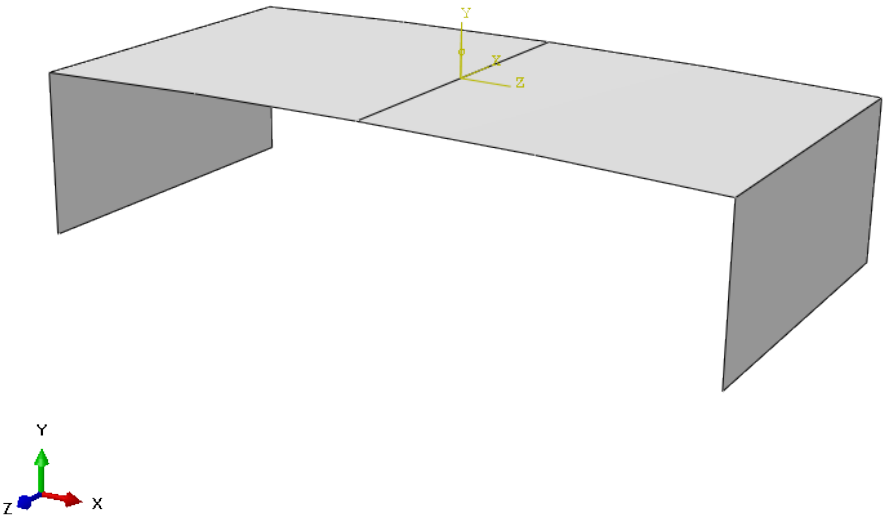
4. The **Create Datum CSYS** dialog opens. Name the CSYS **CSYS_VaryingThickness** and select **Coordinate system Type** as **Rectangular**. Press **Continue**.



5. In the viewport select the point in the middle of the deck shown in the figure below to be the **Origin**. This will be the origin of the shell thickness variation. Select one of the edge points on the middle of the bridge deck to be on the **X-axis** and click on the previously created datum point to be on the **X-Y plane**.



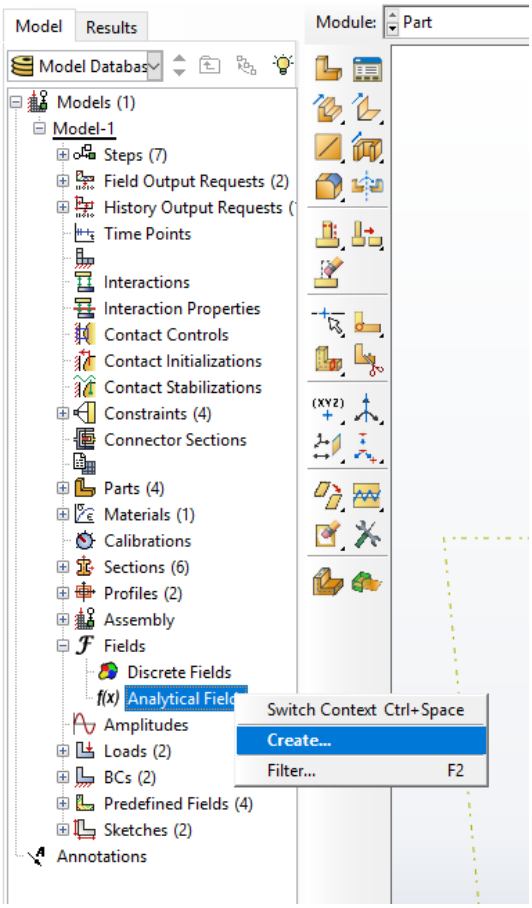
6. Close the **Create Datum CSYS** dialog that opens. The created coordinate system is now shown in the viewport. The model should now look like the figure below.



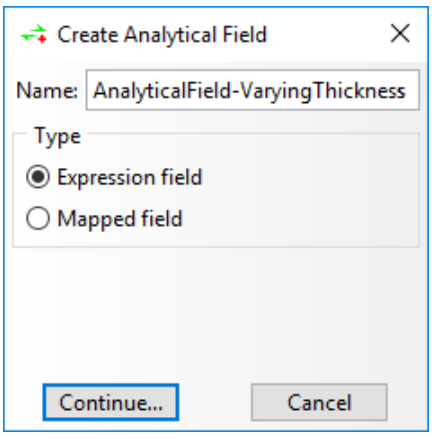
2.3 Define an analytical field

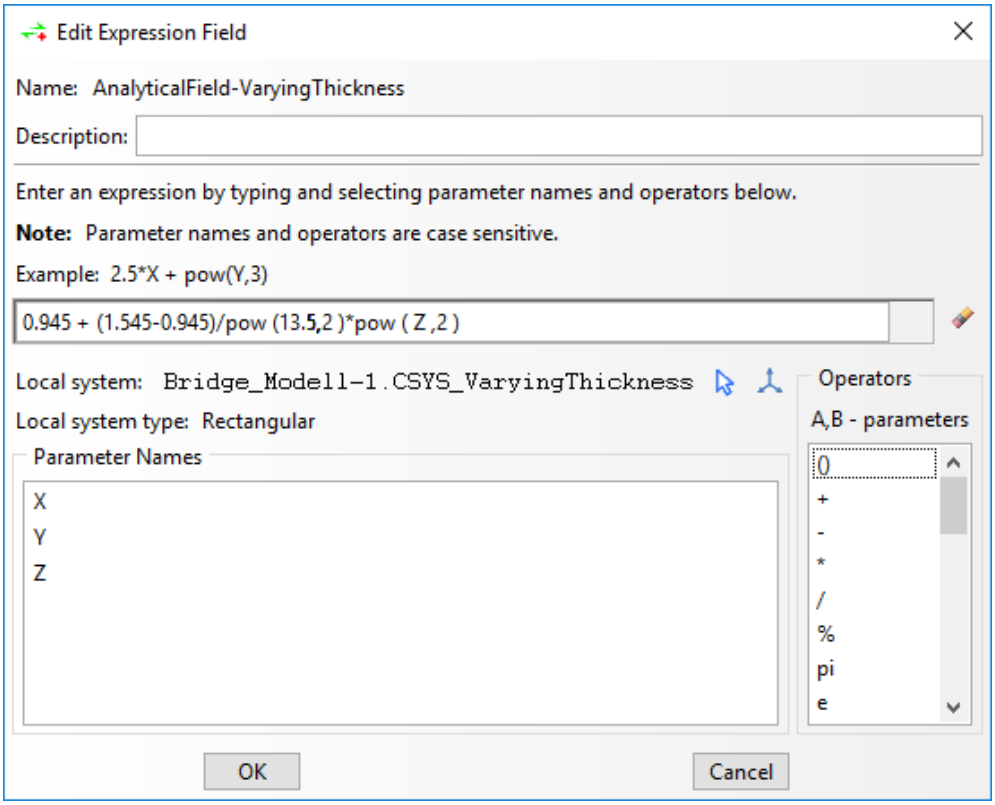
The next step is to define an analytical field using the previously created CSYS.

1. **Analytical Fields** are found in the model tree under **Fields**. Press right mouse button and select **Create.**



2. The **Create Analytical field** dialog opens. Name the analytical field **AnalyticalField-VaryingThickness** and select **Expression field**. Press **Continue**. The **Edit Expression Field** dialog opens.





3. Press to select **Local system** in the viewport. Click on the CSYS previously created. Enter the expression $0.945 + (1.545 - 0.945)/pow\ (\ 13.5\ ,2\ ) * pow\ (\ Z\ ,2\ )$ for the varying shell thickness. In this case the thickness will vary from 0.945 m from the middle to 1.545 m at the ends along the local Z-axis to the chosen CSYS. It is possible to use **Parameter Names** and **Operators** from the dialog box. Press **OK**.

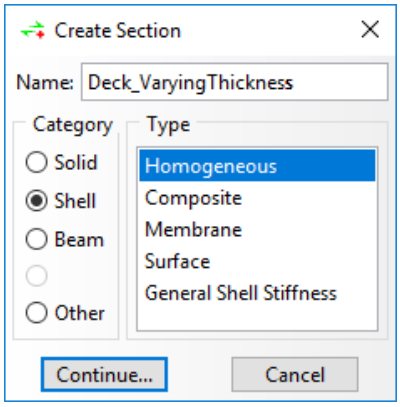
NOTE: The analytical field expression defines how the thickness will vary along a coordinate axis. This means that **Z** in the expression field corresponds to a position along the **Z-axis** of the chosen **CSYS**, **Y** corresponds to the **Y-axis** and so on. For this example, a parabolic variation is chosen. Since it is known that the thickness will vary from 0.945m to 1.545m it is created through a parabolic function, $y = az^2 + b$ where y is the thickness at corresponding length z and b is the thickness at the starting point of the **CSYS**, which for this case is 0.945m. In this example z corresponds to the length along the local **Z-axis** as defined in the previously created **CSYS**. As $z = 13.5$ when $y = 1.545$ it results in $a = \frac{(1.545-0.945)}{13.5^2}$

2.4 Create and assign shell section

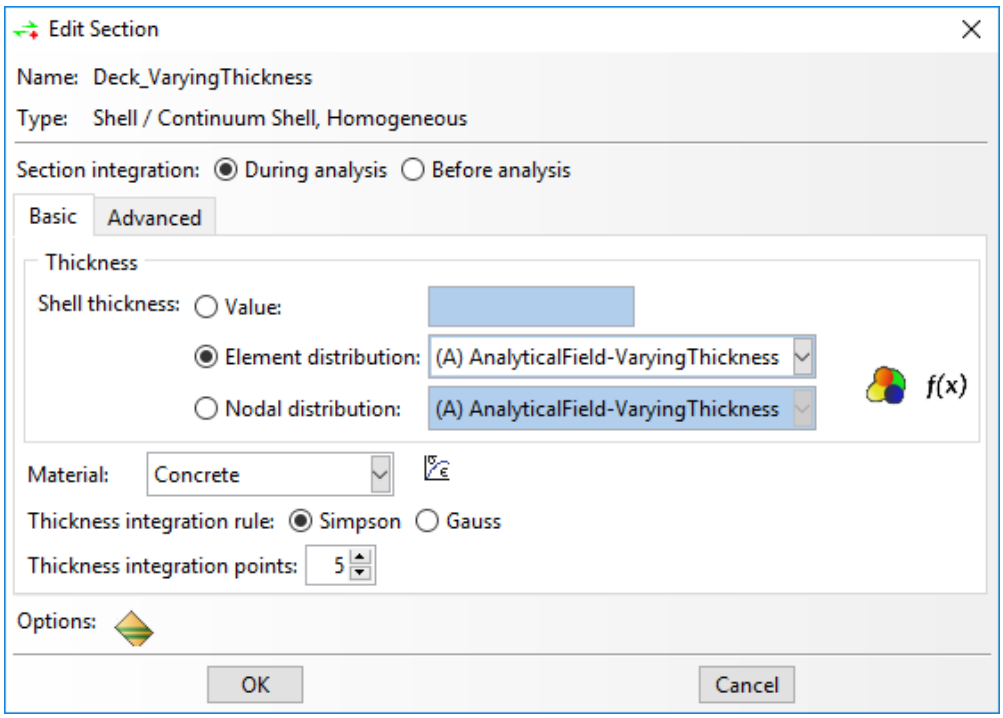
The next step is to create and assign shell sections.

1. Switch to the **Property** module and select **Section → Create** from the main menu or press the **Create Section** icon  .

2. The **Create Section** dialog opens. Name the section **Deck_VaryingThickness**. Set the **Category** radio button to **Shell** and choose **Homogeneous** in the **Type** list. Press **Continue.**



3. The **Edit Section** dialog opens. Choose **Element distribution** and select **AnalyticalField-VaryingThickness** in the drop down list.

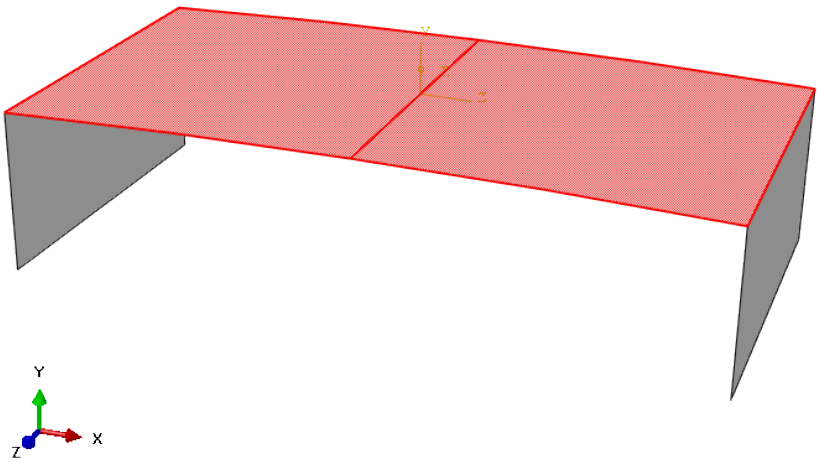


NOTE: For this tutorial **Element distribution** is used for the varying shell thickness. The difference between **Nodal distribution** and **Element distribution** is that for **Nodal distribution** the thickness is interpolated from the calculated thickness of the corresponding nodes for the element i.e. one thickness is calculated for each node from which the element thickness is interpolated from. For **Element distribution** one thickness is calculated for the element. The thickness is calculated as the mean value of the nodal thicknesses for the element. For this example with a structured mesh, a linear thickness variation and reduced integration there will be no difference between these options. Since the calculation of the shell thickness is mesh dependent a refined mesh will render a thickness closer to the real value.

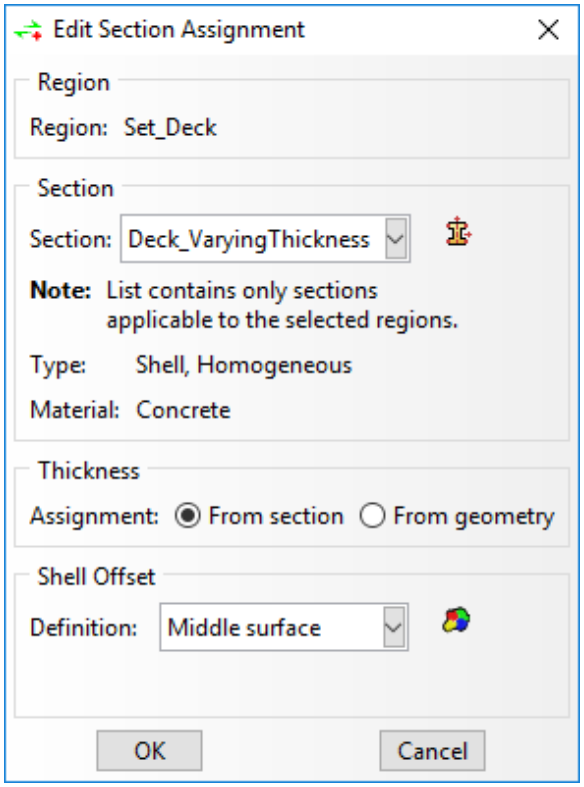
4. Click **OK** to close the dialog box.

NOTE: In the **Edit Section** dialog box you are able to create analytical expression field by pressing $f(x)$.

5. The next step is to assign the created sections to the bridge deck. From the main menu select **Assign → Section**. Name the set **Set_Deck** and select the whole deck in the viewport. Press **Done** in the prompt area.

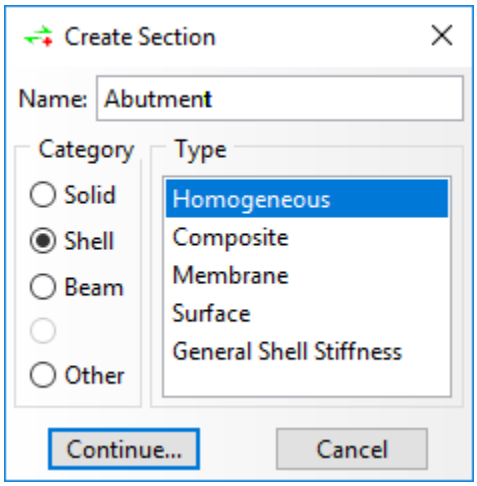


6. The **Edit Section Assignment** dialog opens. Select **Deck_VaryingThickness** in the **Section** drop-down list and **Middle surface** in the **Shell Offset Definition** drop-down list. Press **OK**.

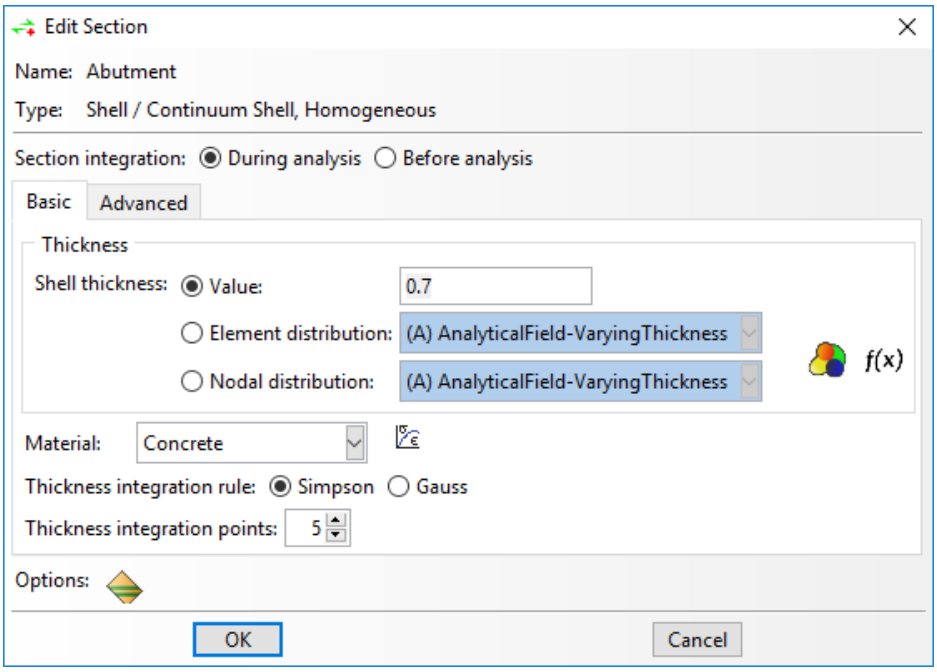


NOTE: Section forces are calculated around the reference plane for the shell section. If section forces are to be obtained around the neutral axis one should always use **Middle surface** as **Shell Offset Definition**.

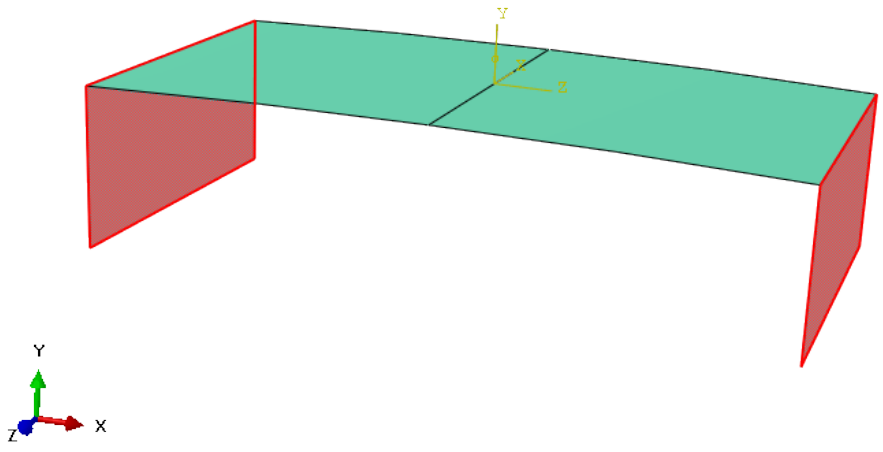
7. Finally the abutments has to be assigned a section. Repeat step 1-2 and name the section **Abutment**.



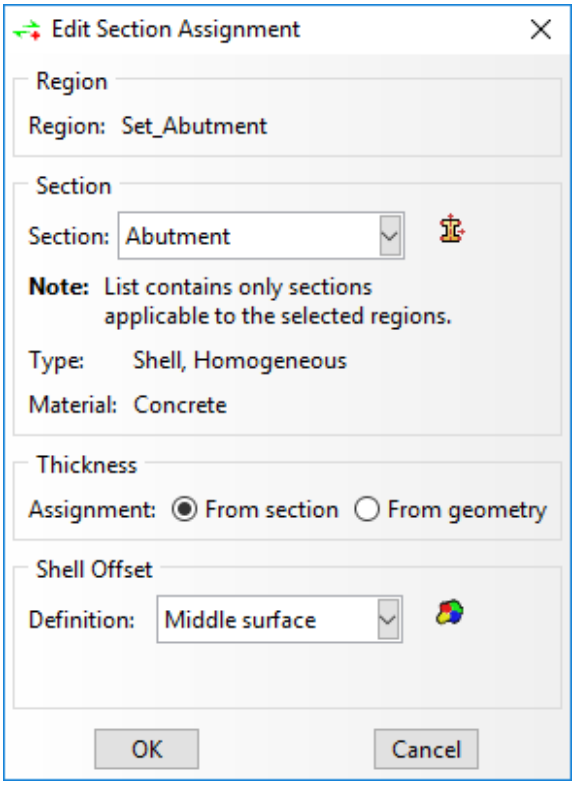
8. In the **Edit Section** dialog that opens under **Thickness** choose the **Value** radio button and enter a thickness of 0.7m since the abutment thickness is constant. Press **OK**.



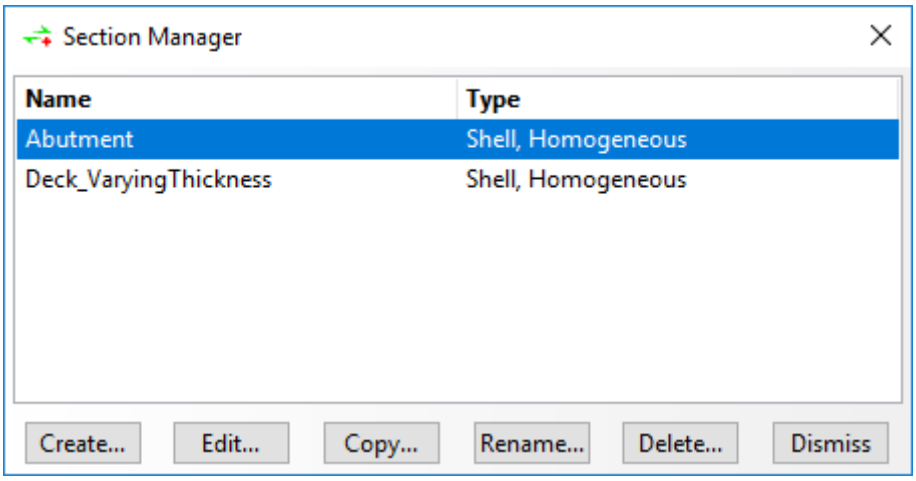
9. Repeat step 5-6 and name the set **Set_Abutment**. Select the two abutments in the viewport and press **Done** in the prompt area.



10. In the **Edit Section Assignment** dialog that opens select the previously created section **Abutment** in the **Section** drop-down list and select **Middle surface** in the **Shell Offset Definition** drop-down list. Press **OK**.



11. The two created sections should now be present in the **Section Manager**.

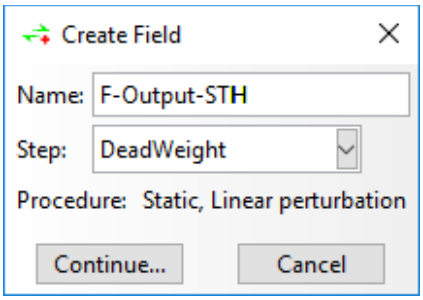


3 Creating output field request

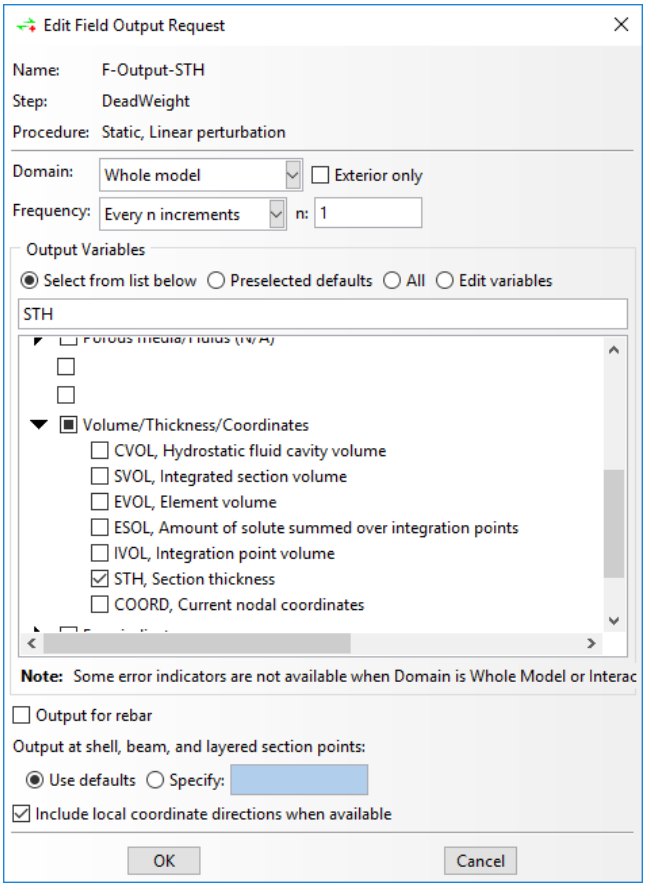
In order to be able to visualize the varying shell thickness the field output variable **STH** needs to be requested.

1. Switch to the **Step** module. From the main menu select **Output → Field Output Requests → Create**.

2. In the **Create Field** dialog, name the field output **F-Output-STH** and select **DeadWeight** in the **Step** drop-down list. Press **Continue**.



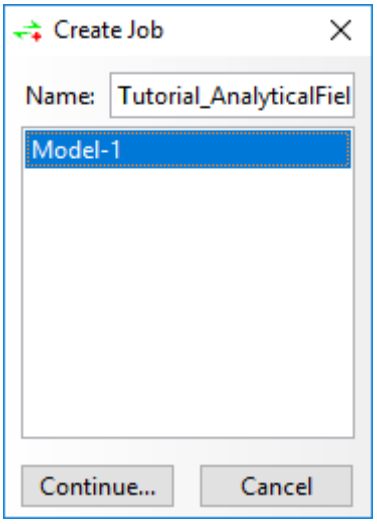
3. The **Edit Field Output Request** dialog opens. Under **Output Variables** choose **Volume/Thickness/Coordinates** and **STH, Section thickness** according to the figure below. Press **OK**.



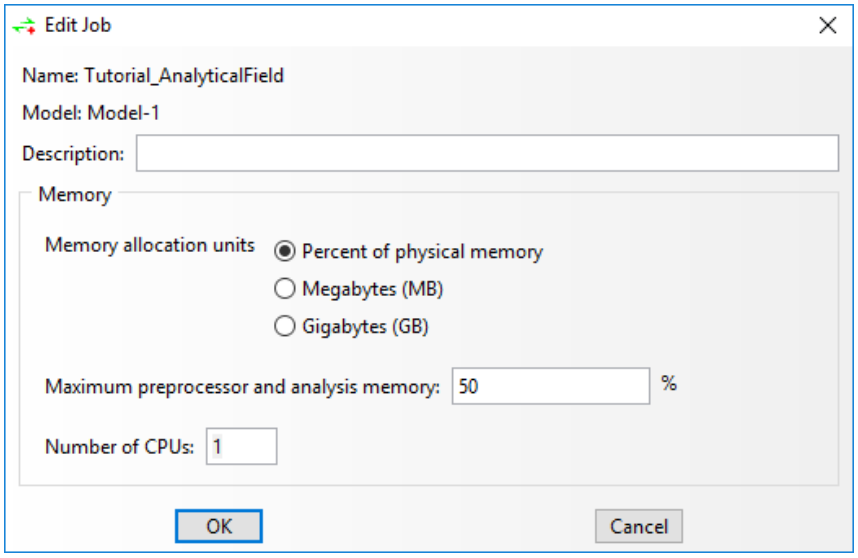
4 Visualizing the varying shell thickness

It is not possible to render a varying shell thickness. Instead a contour plot of the field output **STH** is used to verify the varying thickness after the analysis.

1. Switch to the **Job** module, from the main menu select **Job → Create** or press the icon  . The **Create Job** dialog opens, name the job **Tutorial_AnalyticalField**. Press **Continue**.

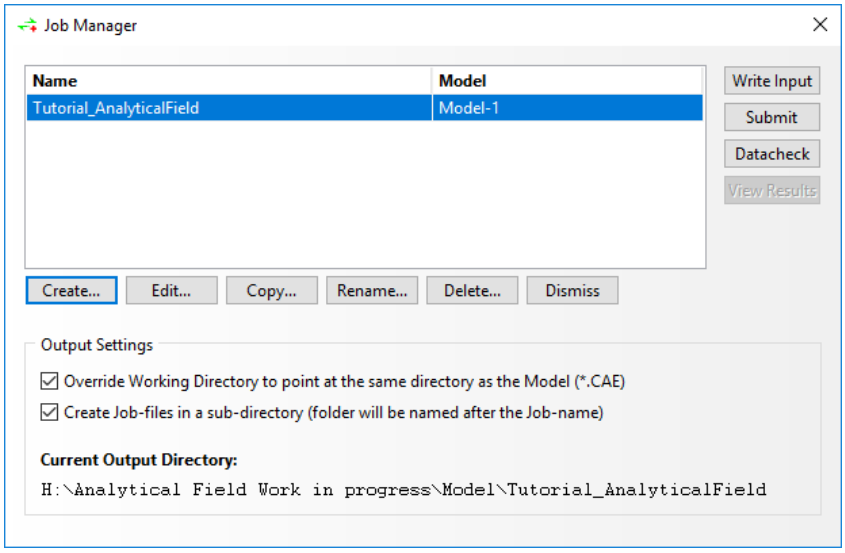


2. In the **Edit Job** dialog press **OK**.



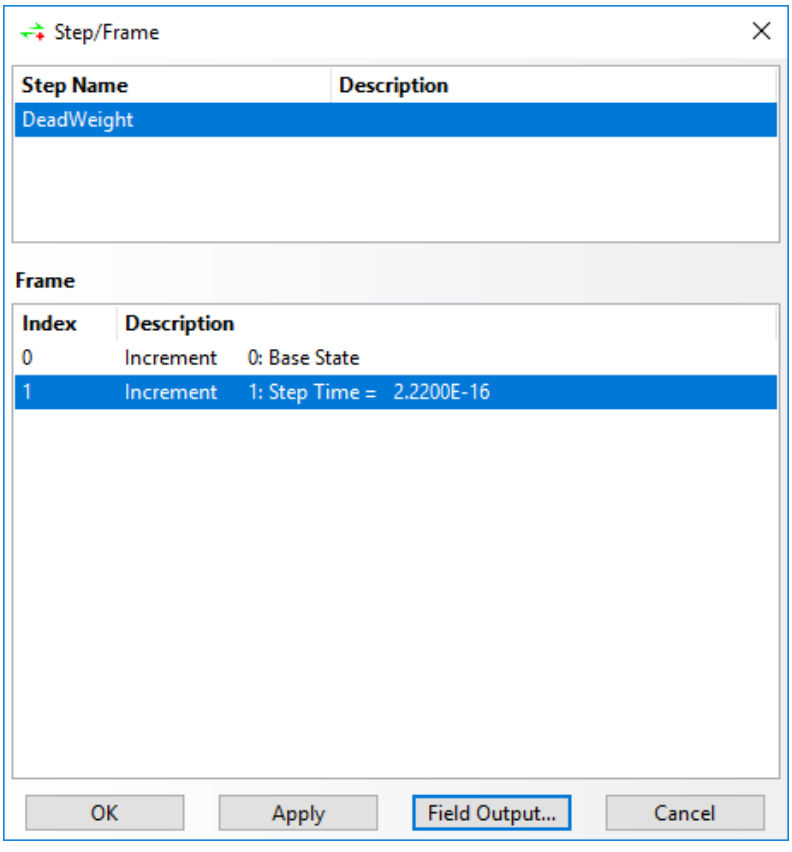
3. From the main menu, select **Job → Manager** or press the icon  .

4. Select the previously created job and press **Submit**.

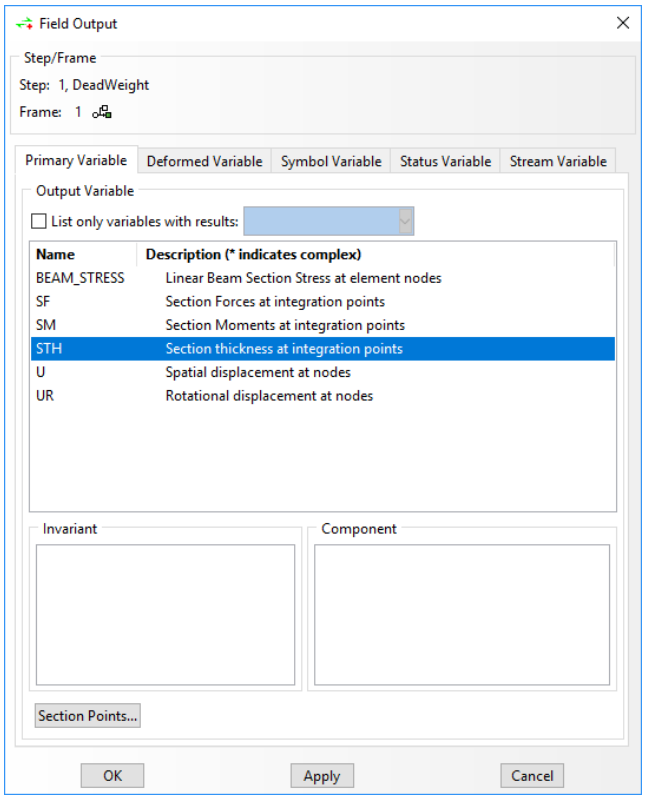


5. When the job is completed select **View Results** in the **Job Manager** dialog and the program will directly switch to the **Visualization** module.

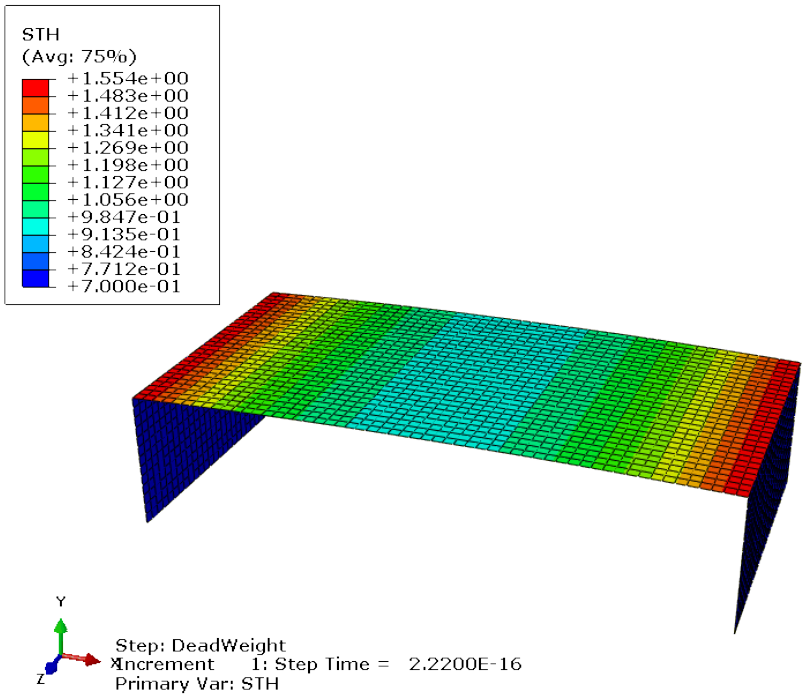
6. From the main menu, select **Result → Step/Frame**. In the **Step/Frame** dialog, select **DeadWeight** and **Frame 1**. Press **Apply** and then press **Field Output**.



7. In the **Field Output** dialog, select **STH** and press **Apply**. Press **OK** in both dialogs to close them.

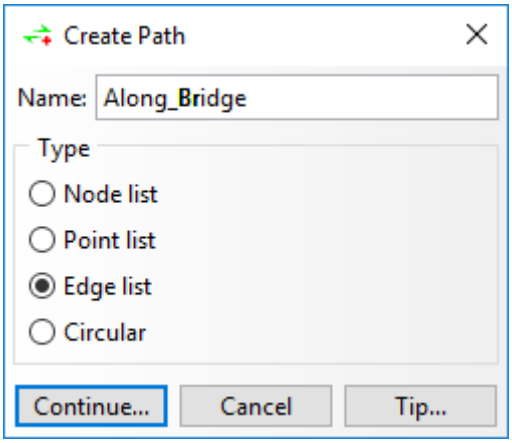


8. To visualize the thickness, from the main menu select **Plot → Contour → On Undeformed Shape.** The contours in the figure below shows the varying shell thickness.

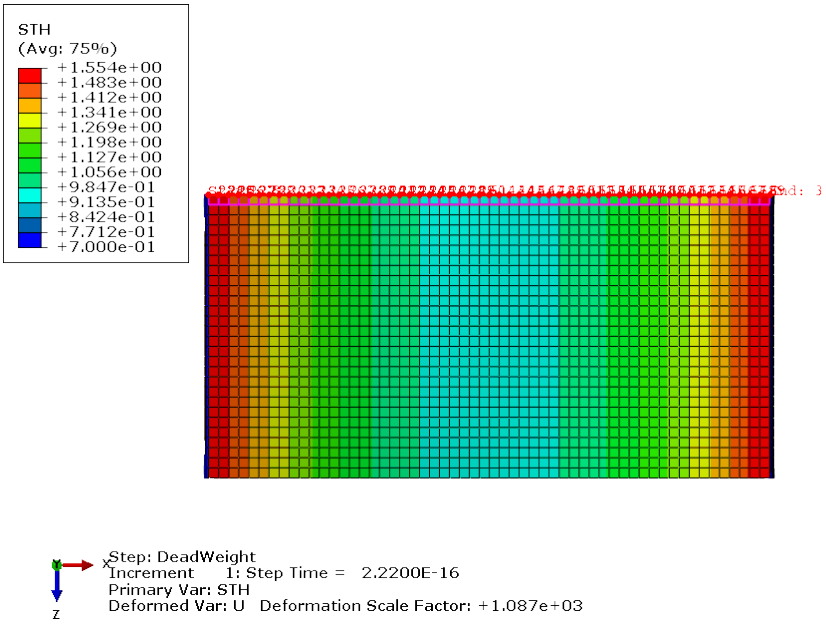


To check the thickness in the different sections it is possible to use **Probe Values** or **XY Data**. For this tutorial **XY Data** will be used in order to create a plot of the thickness along the bridge deck. Begin with creating a **Path** through a line along the bridge.

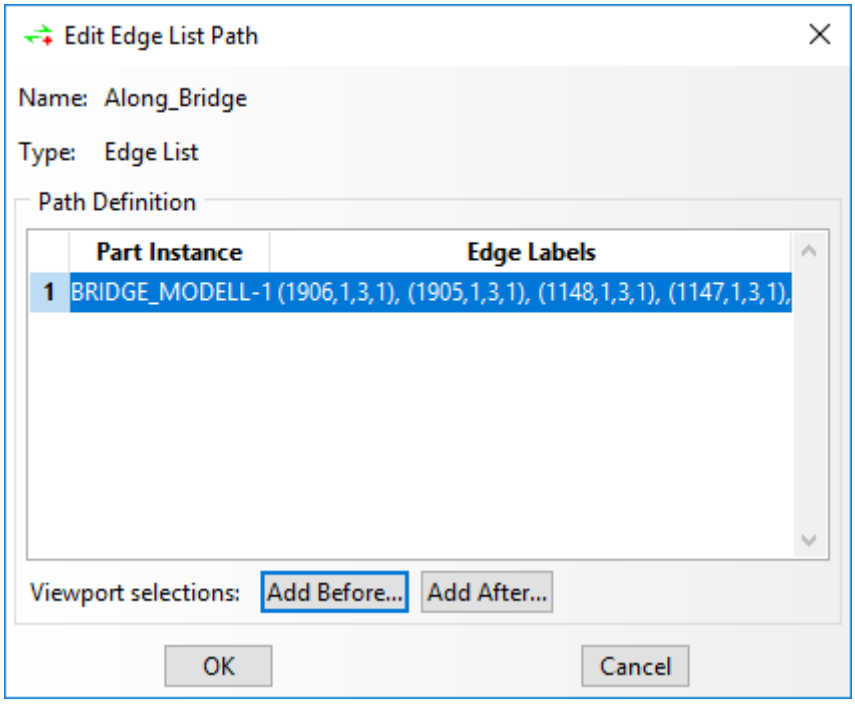
1. From the main menu select, **Tools → Paths → Create**. The **Create Path** dialog opens. Name the path **Along_Bridge** Select **Edge list** under **Type** and press **Continue**.



2. The **Edit Edge List Path** dialog opens. Press **Add Before…** and select an element on the end for one of the edges of the bridge deck in the viewport according to the figure below. Press **Done** in the prompt area.

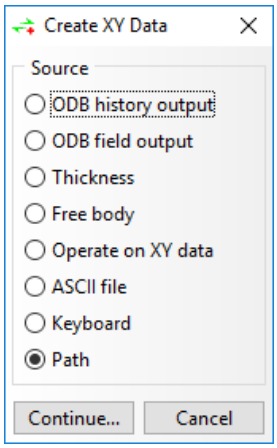


3. The **Edit Edge List Path** dialog opens again. Press **OK**.

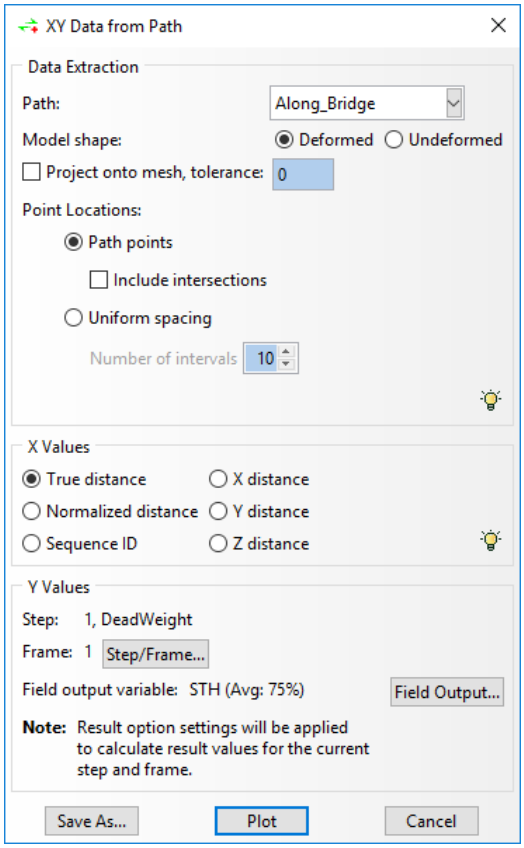


The next step is to create an XY-plot over the varying thickness.

4. From the main menu select **Tools → XY Data → Create**. The **Create XY Data** dialog opens, select **Path** under **Source** and press **Continue**.



5. The **XY Data from Path** dialog opens. Select the created path for the bridge in the **Path** drop-down list. Under **X Values** choose **X distance**. Under **Y Values** choose **STH** as **Field output variable**. Press **Plot**.



6. A plot showing the how the thickness varies along the bridge will then appear in the viewport according to the figure below. From this it is possible to control the thickness along the bridge.

Figure 4-1 XY-Data plot of the varying thickness along the bridge

In the diagram below an illustration is made between the thickness calculated by BRIGADE/Plus and the thickness stated in Figure 2-1. The blue line shows the same data as in Figure 4-1. As can be seen in the figure, the applied thickness variation corresponds well to the thickness in Figure 2-1.

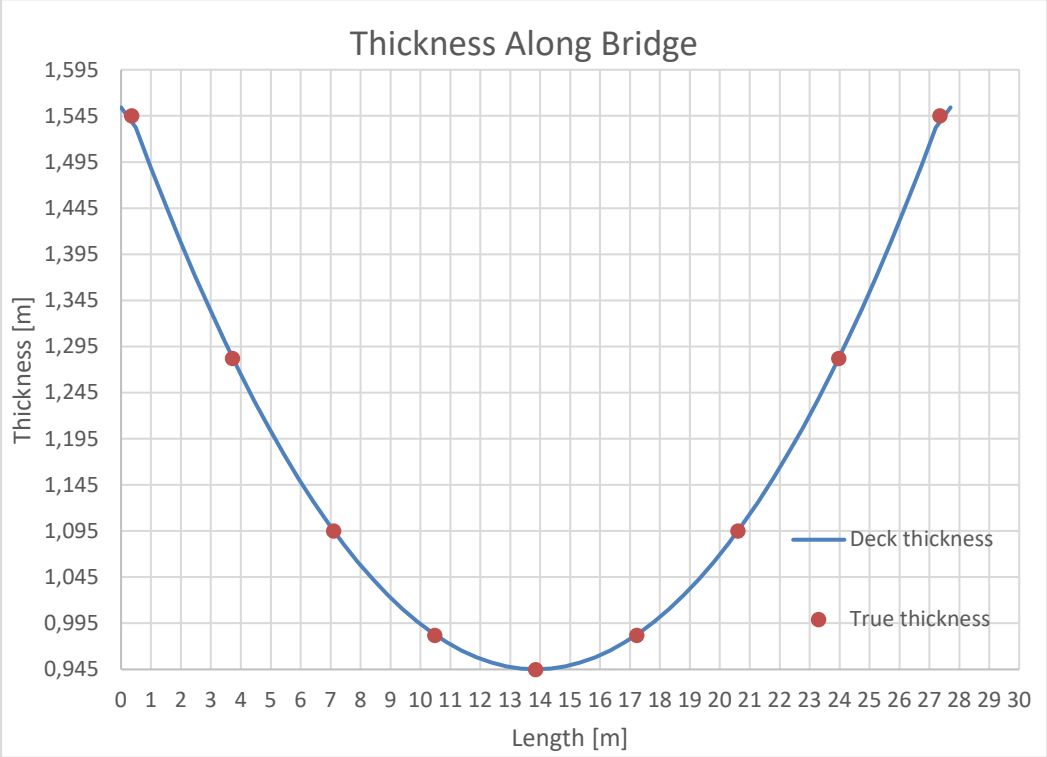


Figure 4-2 Comparison between STH data and the thickness from Figure 2-1.