

# Joint-pattern first steps

Tutorial	
Name:	Joint-pattern first steps
Description:	This tutorial provides an introduction to the assignment of joint patterns.
Program:	SAP2000
Version:	12.1.0
Model ID:	na

To obtain the vertical loading on a **slab** element which is modeled using a fairly fine **mesh**, users may implement a **joint pattern** through the following process:

- For each loading patch, or set of patches which act together, select Define > Joint Pattern to define a **joint pattern**, as shown in Figure 1:

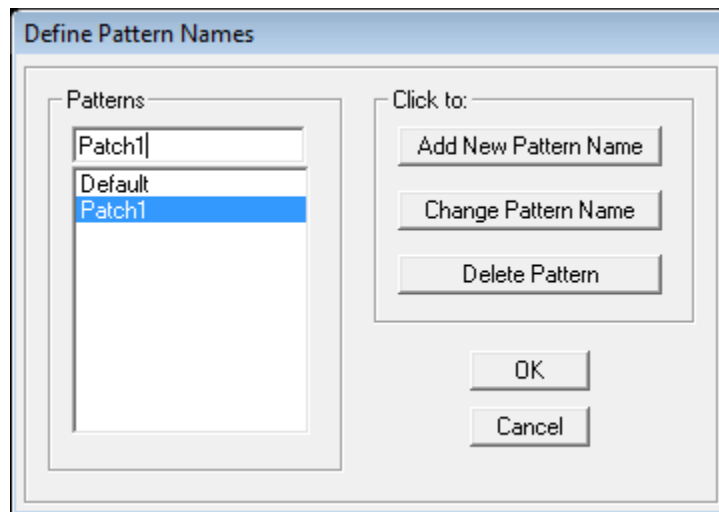


Figure 1 - Name joint patterns

- Select joints which are contained within the patch or patches. Next, select Assign > Joint Pattern, then assign a dimensionless scale factor  $D$ , such as 1.0. This process is shown in Figure 2:

**Pattern Data**

**Pattern Name** Patch1

**Pattern Assignment Type**

X, Y, Z Multipliers (Pattern Value = Ax + By + Cz + D)

Z Coordinate at Zero Pressure and Weight Per Unit Volume

Pattern Value = Ax + By + Cz + D

Constant A 0.

Constant B 0.

Constant C 0.

Constant D 1.

**Restrictions**

Use all values

Zero Negative values

Zero Positive values

**Options**

Add to existing values

Replace existing values

Delete existing values

OK Cancel

Figure 2 - Pattern data

- Select the slab elements, then select Assign > Area Loads > Surface Pressure to assign pressure (force/area) such that it acts downward on the top face, according to the appropriate [load pattern](#). This menu is shown in Figure 3:

**Area Surface Pressure Load**

Load Pattern Name PatchLoad1

Units KN, m, C

**Pressure**

By Element

Pressure

By Joint Pattern

Pattern Patch1

Multiplier 5

**Face**

Top

**Options**

Add to Existing Loads

Replace Existing Loads

Delete Existing Loads

OK Cancel

Figure 3 - Area surface pressure load

- The pressure on each element will be interpolated from the product between joint-pattern values and the pressure assigned, which is 5.0 kN/m in this example. If only one element joint has a nonzero pressure, a quarter of the load will be applied. Load pressure is multiplied by element tributary area such that users need not manually calculate nodal forces.

## Display joint pattern

- To graphically display the [joint-pattern](#) assignment, select Display > Show Misc Assigns > Joints.
- Select Display > Show Load Assign > Area Surface Pressure Values to display the area load assigned to each [joint](#).